ANSYS Fluent User's Guide

ANSYS, Inc.
Southpointe
2600 ANSYS Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 18.1
April 2017

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2008 companies.

Copyright and Trademark Information

© 2017 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

ANSYS, ANSYS Workbench, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners. FLEMlm and FLEXnet are trademarks of Flexera Software LLC.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2008 companies.

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the legal information in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, contact ANSYS, Inc.

Published in the U.S.A.

Table of Contents

Using This Manual lxv
1. The Contents of This Manual lxv
2. The Contents of the Fluent Manuals lxvii
3. Typographical Conventions lxix
4. Mathematical Conventions lxxi
5. Technical Support lxxii
1. Graphical User Interface (GUI) 1
1.1. GUI Components 1
1.1.1. The Ribbon 2
1.1.2. The Tree 3
1.1.3. Graphics Windows 3
1.1.4. Toolbars 7
1.1.4.1. The Standard Toolbar 7
1.1.4.2. The Graphics Toolbar 7
1.1.4.2.1. Pointer Tools 7
1.1.4.2.2. View Tools 8
1.1.4.2.3. Projection Tools 9
1.1.4.2.4. Display Options 9
1.1.5. Task Pages 10
1.1.6. The Console 10
1.1.7. Dialog Boxes 11
1.1.7.1. Input Controls 13
1.1.7.1.1. Tabs 13
1.1.7.1.2. Buttons 14
1.1.7.1.3. Check Boxes 14
1.1.7.1.4. Radio Buttons 14
1.1.7.1.5. Text Entry Boxes 14
1.1.7.1.6. Integer Number Entry Boxes 14
1.1.7.1.7. Real Number Entry Boxes 15
1.1.7.1.8. Filter Text Entry Boxes 15
1.1.7.1.9. Single-Selection Lists 15
1.1.7.1.10. Multiple-Selection Lists 16
1.1.7.1.11. Drop-Down Lists 16
1.1.7.1.12. Scales 17
1.1.7.2. Types of Dialog Boxes 17
1.1.7.2.1. Information Dialog Boxes 17
1.1.7.2.2. Warning Dialog Boxes 18
1.1.7.2.3. Error Dialog Boxes 18
1.1.7.2.4. The Working Dialog Box 18
1.1.7.2.5. Question Dialog Box 18
1.1.7.2.6. The Select File Dialog Box 19
1.1.7.2.6.1. The Select File Dialog Box (Windows) 19
1.1.7.2.6.2. The Select File Dialog Box (Linux) 19
1.2. Customizing the Graphical User Interface 22
1.3. Using the GUI Help System 23
1.3.1. Task Page and Dialog Box Help 23
1.3.2. Context-Sensitive Help (Linux Only) 23
1.3.3. Opening the User's Guide Table of Contents 24
1.3.4. Opening the Reference Guide 25
1.3.5. Help on Help 25

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.3.6. Accessing Printable (PDF) Manuals 25
1.3.7. Help for Text Interface Commands 25
1.3.8. Accessing the Customer Portal Web Site 26
1.3.9. Obtaining License Use Information 26
1.3.10. Version and Release Information 26
2. Text User Interface (TUI) 27
2.1. Text Menu System 27
2.1.1. Command Abbreviation 28
2.1.2. Command Line History 29
2.1.3. Scheme Evaluation 29
2.1.4. Aliases 30
2.2. Text Prompt System 30
2.2.1. Numbers 31
2.2.2. Booleans 31
2.2.3. Strings 31
2.2.4. Symbols 31
2.2.5. Filenames 31
2.2.6. Lists 32
2.2.7. Evaluation 33
2.2.8. Default Value Binding 34
2.3. Interrupts 34
2.4. System Commands 34
2.4.1. System Commands for Linux-based Operating Systems 34
2.4.2. System Commands for Windows Operating Systems 35
2.5. Text Menu Input from Character Strings 35
2.6. Using the Text Interface Help System 36
3. Reading and Writing Files 39
3.1. Shortcuts for Reading and Writing Files 39
3.1.1. Default File Suffixes 40
3.1.2. Binary Files 41
3.1.3. Detecting File Format 41
3.1.4. Recent File List 41
3.1.5. Reading and Writing Compressed Files 41
3.1.5.1. Reading Compressed Files 41
3.1.5.2. Writing Compressed Files 42
3.1.6. Tilde Expansion (Linux Systems Only) 43
3.1.7. Automatic Numbering of Files 43
3.1.8. Disabling the Overwrite Confirmation Prompt 44
3.1.9. Toolbar Buttons 44
3.2. Reading Mesh Files 44
3.3. Reading and Writing Case and Data Files 45
3.3.1. Reading and Writing Case Files 46
3.3.2. Reading and Writing Data Files 47
3.3.3. Reading and Writing Case and Data Files Together 47
3.3.4. Reading and Writing Files Using Hierarchical Data Format (HDF) 47
3.3.5. Automatic Saving of Case and Data Files 48
3.4. Reading and Writing Parallel Data Files 51
3.4.1. Writing Parallel Data Files 52
3.4.2. Reading Parallel Data Files 52
3.4.3. Availability and Limitations 52
3.5. Reading Fluent/UNS and RAMPANT Case and Data Files 53
3.6. Reading and Writing Profile Files 53

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.6.1. Reading Profile Files 54
3.6.2. Writing Profile Files 54
3.7. Reading and Writing Boundary Conditions 56
3.8. Writing a Boundary Mesh 56
3.9. Reading Scheme Source Files 57
3.10. Creating and Reading Journal Files 57
3.10.1. Procedure 59
3.11. Creating Transcript Files 59
3.12. Importing Files 59
3.12.1. ABAQUS Files 62
3.12.2. CFX Files 62
3.12.3. Meshes and Data in CGNS Format 63
3.12.4. EnSight Files 64
3.12.5. ANSYS FIDAP Neutral Files 64
3.12.6. GAMBIT and GeoMesh Mesh Files 64
3.12.7. HYPERMESH ASCII Files 65
3.12.8. I-deas Universal Files 65
3.12.9. LSTC Files 65
3.12.10. Marc POST Files 65
3.12.11. Mechanical APDL Files 66
3.12.12. NASTRAN Files 66
3.12.13. PATRAN Neutral Files 67
3.12.14. PLOT3D Files 67
3.12.15. PTC Mechanica Design Files 67
3.12.16. Tecplot Files 67
3.12.17. Fluent 4 Case Files 68
3.12.18. PreBFC Files 68
3.12.19. Partition Files 68
3.12.20. CHEMKIN Mechanism 68
3.13. Exporting Solution Data 69
3.13.1. Exporting Limitations 69
3.14. Exporting Solution Data after a Calculation 70
3.14.1. ABAQUS Files 71
3.14.2. Mechanical APDL Files 72
3.14.3. Mechanical APDL Input Files 73
3.14.4. ASCII Files 74
3.14.5. AVS Files 74
3.14.6. ANSYS CFD-Post-Compatible Files 74
3.14.7. CGNS Files 75
3.14.8. Data Explorer Files 76
3.14.9. EnSight Case Gold Files 76
3.14.10. FAST Files 79
3.14.11. FAST Solution Files 80
3.14.12. FieldView Unstructured Files 80
3.14.13. I-deas Universal Files 81
3.14.14. NASTRAN Files 82
3.14.15. PATRAN Files 82
3.14.16. RadTherm Files 83
3.14.17. Tecplot Files 83
3.15. Exporting Steady-State Particle History Data 84
3.16. Exporting Data During a Transient Calculation 86
3.16.1. Creating Automatic Export Definitions for Solution Data 87

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.16.2. Creating Automatic Export Definitions for Transient Particle History Data 89
3.17. Exporting to ANSYS CFD-Post 91
3.18. Managing Solution Files 93
3.19. Mesh-to-Mesh Solution Interpolation 94
3.19.1. Performing Mesh-to-Mesh Solution Interpolation 94
3.19.2. Format of the Interpolation File 96
3.20. Mapping Data for Fluid-Structure Interaction (FSI) Applications 97
3.20.1. FEA File Formats 97
3.20.2. Using the FSI Mapping Dialog Boxes 97
3.21. Saving Picture Files 102
3.21.1. Using the Save Picture Dialog Box 102
3.21.1.1. Choosing the Picture File Format 103
3.21.1.2. Specifying the Color Mode 105
3.21.1.3. Choosing the File Type 105
3.21.1.4. Defining the Resolution 105
3.21.1.5. Picture Options 105
3.21.2. Picture Options for PostScript Files 106
3.21.2.1. Window Dumps (Linux Systems Only) 106
3.21.2.2. Previewing the Picture Image 107
3.22. Setting Data File Quantities 107
3.23. The .fluent File 108
4. Unit Systems 111
4.1. Restrictions on Units 111
4.2. Units in Mesh Files 112
4.3. Built-In Unit Systems in ANSYS Fluent 112
4.4. Customizing Units 112
4.4.1. Listing Current Units 113
4.4.2. Changing the Units for a Quantity 113
4.4.3. Defining a New Unit 113
4.4.3.1. Determining the Conversion Factor 114
5. Reading and Manipulating Meshes 115
5.1. Mesh Topologies 115
5.1.1. Examples of Acceptable Mesh Topologies 116
5.1.2. Face-Node Connectivity in ANSYS Fluent 121
5.1.2.1. Face-Node Connectivity for Triangular Cells 122
5.1.2.2. Face-Node Connectivity for Quadrilateral Cells 123
5.1.2.3. Face-Node Connectivity for Tetrahedral Cells 124
5.1.2.4. Face-Node Connectivity for Wedge Cells 125
5.1.2.5. Face-Node Connectivity for Pyramidal Cells 126
5.1.2.6. Face-Node Connectivity for Hex Cells 127
5.1.2.7. Face-Node Connectivity for Polyhedral Cells 128
5.1.3. Choosing the Appropriate Mesh Type 128
5.1.3.1. Setup Time 128
5.1.3.2. Computational Expense 129
5.1.3.3. Numerical Diffusion 129
5.2. Mesh Requirements and Considerations 130
5.2.1. Geometry/Mesh Requirements 130
5.2.2. Mesh Quality 131
5.2.2.1. Mesh Element Distribution 133
5.2.2.2. Cell Quality 134
5.2.2.3. Smoothness 135
5.2.2.4. Flow-Field Dependency 135

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.3. Mesh Sources 135
5.3.1. ANSYS Meshing Mesh Files 135
5.3.2. Fluent Meshing Mode Mesh Files 136
5.3.3. Fluent Meshing Mesh Files 136
5.3.4. GAMBIT Mesh Files 136
5.3.5. GeoMesh Mesh Files 136
5.3.6. PreBFC Mesh Files 136
5.3.6.1. Structured Mesh Files 136
5.3.6.2. Unstructured Triangular and Tetrahedral Mesh Files 137
5.3.7. ICEM CFD Mesh Files 137
5.3.8. I-deas Universal Files 137
5.3.8.1. Recognized I-deas Datasets 138
5.3.8.2. Grouping Nodes to Create Face Zones 138
5.3.8.3. Grouping Elements to Create Cell Zones 138
5.3.8.4. Deleting Duplicate Nodes 138
5.3.9. NASTRAN Files 138
5.3.9.1. Recognized NASTRAN Bulk Data Entries 139
5.3.9.2. Deleting Duplicate Nodes 139
5.3.10. PATRAN Neutral Files 139
5.3.10.1. Recognized PATRAN Datasets 140
5.3.10.2. Grouping Elements to Create Cell Zones 140
5.3.11. Mechanical APDL Files 140
5.3.11.1. Recognized ANSYS 5.4 and 5.5 Datasets 141
5.3.12. CFX Files 141
5.3.13. Using the fe2ram Filter to Convert Files 142
5.3.14. Using the `tpoly` Filter to Remove Hanging Nodes/Edges 143
5.3.14.1. Limitations 144
5.3.15. Fluent/UNS and RAMPANT Case Files 144
5.3.16. FLUENT 4 Case Files 144
5.3.17. ANSYS FIDAP Neutral Files 145
5.3.18. Reading Multiple Mesh/Case/Data Files 145
5.3.18.1. Reading Multiple Mesh Files via the Solution Mode of Fluent 146
5.3.18.2. Reading Multiple Mesh Files via the Meshing Mode of Fluent 147
5.3.18.3. Reading Multiple Mesh Files via tmerge 148
5.3.19. Reading Surface Mesh Files 149
5.4. Non-Conformal Meshes 150
5.4.1. Non-Conformal Mesh Calculations 150
5.4.1.1. The Periodic Boundary Condition Option 153
5.4.1.2. The Periodic Repeats Option 154
5.4.1.3. The Coupled Wall Option 156
5.4.1.4. Matching Option 157
5.4.1.5. The Mapped Option 158
5.4.1.6. The Static Option 160
5.4.1.7. Interface Zones Automatic Naming Conventions 160
5.4.1.7.1. Default (No Options Enabled) 160
5.4.1.7.2. Periodic Boundary Condition 160
5.4.1.7.3. Periodic Repeats 161
5.4.1.7.4. Coupled Wall 161
5.4.1.7.5. Matching 161
5.4.1.7.6. Mapped 161
5.4.1.7.7. Static 162
5.4.2. Non-Conformal Interface Algorithm 162

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.4.3. Requirements and Limitations of Non-Conformal Meshes 163
5.4.4. Using a Non-Conformal Mesh in ANSYS Fluent 164
5.4.4.1. Manually Creating Mesh Interfaces 170
5.5. Overset Meshes 172
5.5.1. Introduction 172
5.5.2. Overset Topologies 174
5.5.3. Overset Domain Connectivity 177
5.5.3.1. Hole Cutting 177
5.5.3.2. Overlap Minimization 178
5.5.3.3. Donor Search 181
5.5.4. Diagnosing Overset Interface Issues 181
5.5.4.1. Flood Filling Fails During Hole Cutting 182
5.5.4.1.1. Incorrect Seed Cells 182
5.5.4.1.2. Leakage Between Overlapping Boundaries 182
5.5.4.2. Donor Search Fails Due to Orphan Cells 182
5.5.5. Overset Meshing Best Practices 183
5.5.6. Overset Meshing Limitations and Compatibilities 184
5.5.6.1. Limitations 184
5.5.6.2. Compatibilities 185
5.5.7. Setting up an Overset Interface 186
5.5.8. Postprocessing Overset Meshes 187
5.5.8.1. Overset Mesh Display 188
5.5.8.2. Overset Field Functions 189
5.5.8.3. Overset Cell Marks 191
5.5.8.4. Overset Interface listing 191
5.5.8.5. Overset Postprocessing Limitations 191
5.5.9. Writing and Reading Overset Files 191
5.6. Checking the Mesh 192
5.6.1. Mesh Check Report 193
5.6.2. Repairing Meshes 194
5.7. Reporting Mesh Statistics 197
5.7.1. Mesh Size 197
5.7.2. Memory Usage 198
5.7.2.1. Linux Systems 198
5.7.2.2. Windows Systems 198
5.7.3. Mesh Zone Information 199
5.7.4. Partition Statistics 199
5.8. Converting the Mesh to a Polyhedral Mesh 199
5.8.1. Converting the Domain to a Polyhedra 199
5.8.1.1. Limitations 203
5.8.2. Converting Skewed Cells to Polyhedra 204
5.8.2.1. Limitations 204
5.8.3. Converting Cells with Hanging Nodes / Edges to Polyhedra 205
5.8.3.1. Limitations 205
5.9. Modifying the Mesh 206
5.9.1. Merging Zones 206
5.9.1.1. When to Merge Zones 207
5.9.1.2. Using the Merge Zones Dialog Box 207
5.9.2. Separating Zones 208
5.9.2.1. Separating Face Zones 208
5.9.2.1.1. Methods for Separating Face Zones 208
5.9.2.1.2. Inputs for Separating Face Zones 209

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.9.2.2. Separating Cell Zones 210
5.9.2.2.1. Methods for Separating Cell Zones 210
5.9.2.2.2. Inputs for Separating Cell Zones 211
5.9.3. Fusing Face Zones 212
5.9.3.1. Inputs for Fusing Face Zones 213
5.9.3.1.1. Fusing Zones on Branch Cuts 214
5.9.4. Creating Conformal Periodic Zones 214
5.9.5. Slitting Periodic Zones 215
5.9.6. Slitting Face Zones 215
5.9.6.1. Inputs for Slitting Face Zones 216
5.9.7. Orienting Face Zones 217
5.9.8. Extruding Face Zones 217
5.9.8.1. Specifying Extrusion by Displacement Distances 217
5.9.8.2. Specifying Extrusion by Parametric Coordinates 218
5.9.9. Replacing, Deleting, Deactivating, and Activating Zones 218
5.9.9.1. Replacing Zones 218
5.9.9.2. Deleting Zones 219
5.9.9.3. Deactivating Zones 220
5.9.9.4. Activating Zones 221
5.9.10. Copying Cell Zones 222
5.9.11. Replacing the Mesh 222
5.9.11.1. Inputs for Replacing the Mesh 223
5.9.11.2. Limitations 224
5.9.12. Managing Adjacent Zones 224
5.9.12.1. Renaming Zones Using the Adjacency Dialog Box 225
5.9.13. Reordering the Domain 226
5.9.14. Scaling the Mesh 226
5.9.14.1. Using the Scale Mesh Dialog Box 227
5.9.14.1.1. Changing the Unit of Length 227
5.9.14.1.2. Unscaling the Mesh 227
5.9.14.1.3. Changing the Physical Size of the Mesh 228
5.9.15. Translating the Mesh 228
5.9.15.1. Using the Translate Mesh Dialog Box 228
5.9.16. Rotating the Mesh 229
5.9.16.1. Using the Rotate Mesh Dialog Box 229
5.9.17. Improving the Mesh by Smoothing and Swapping 230
5.9.17.1. Smoothing 230
5.9.17.1.1. Quality-Based Smoothing 231
5.9.17.1.2. Laplacian Smoothing 231
5.9.17.1.3. Skewness-Based Smoothing 234
5.9.17.2. Face Swapping 235
5.9.17.2.1. Triangular Meshes 235
5.9.17.2.2. Tetrahedral Meshes 236
5.9.17.3. Combining Skewness-Based Smoothing and Face Swapping 236
6. Cell Zone and Boundary Conditions 237
6.1. Overview 237
6.1.1. Available Cell Zone and Boundary Types 237
6.1.2. The Cell Zone and Boundary Conditions Task Pages 238
6.1.3. Changing Cell and Boundary Zone Types 239
6.1.4. Setting Cell Zone and Boundary Conditions 241
6.1.5. Copying Cell Zone and Boundary Conditions 242
6.1.6. Changing Cell or Boundary Zone Names 243

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.1.7. Defining Non-Uniform Cell Zone and Boundary Conditions 243
6.1.8. Defining and Viewing Parameters 243
6.1.8.1. Creating a New Parameter 246
6.1.8.2. Working With Advanced Parameter Options 247
6.1.8.2.1. Defining Scheme Procedures With Input Parameters 247
6.1.8.2.2. Defining UDFs With Input Parameters 249
6.1.8.2.3. Using the Text User Interface to Define UDFs and Scheme Procedures With Input Parameters 249
6.1.9. Selecting Cell or Boundary Zones in the Graphics Display 250
6.1.10. Operating and Periodic Conditions 251
6.1.11. Highlighting Selected Boundary Zones 251
6.1.12. Saving and Reusing Cell Zone and Boundary Conditions 252
6.2. Cell Zone Conditions 252
6.2.1. Fluid Conditions 252
6.2.1.1. Inputs for Fluid Zones 253
6.2.1.1.1. Defining the Fluid Material 254
6.2.1.1.2. Defining Sources 255
6.2.1.1.3. Defining Fixed Values 255
6.2.1.1.4. Specifying a Laminar Zone 255
6.2.1.1.5. Specifying a Reaction Mechanism 255
6.2.1.1.6. Specifying the Rotation Axis 255
6.2.1.1.7. Defining Zone Motion 256
6.2.1.1.8. Defining Radiation Parameters 259
6.2.2. Solid Conditions 259
6.2.2.1. Inputs for Solid Zones 259
6.2.2.1.1. Defining the Solid Material 260
6.2.2.1.2. Defining a Heat Source 260
6.2.2.1.3. Defining a Fixed Temperature 260
6.2.2.1.4. Specifying the Rotation Axis 260
6.2.2.1.5. Defining Zone Motion 261
6.2.2.1.6. Defining Radiation Parameters 261
6.2.3. Porous Media Conditions 261
6.2.3.1. Limitations and Assumptions of the Porous Media Model 261
6.2.3.2. Momentum Equations for Porous Media 262
6.2.3.2.1. Darcy's Law in Porous Media 263
6.2.3.2.2. Inertial Losses in Porous Media 264
6.2.3.3. Relative Viscosity in Porous Media 264
6.2.3.4. Treatment of the Energy Equation in Porous Media 265
6.2.3.4.1. Equilibrium Thermal Model Equations 265
6.2.3.4.2. Non-Equilibrium Thermal Model Equations 265
6.2.3.5. Treatment of Turbulence in Porous Media 266
6.2.3.6. Effect of Porosity on Transient Scalar Equations 267
6.2.3.7. User Inputs for Porous Media 267
6.2.3.7.1. Defining the Porous Zone 268
6.2.3.7.2. Defining the Porous Velocity Formulation 269
6.2.3.7.3. Defining the Fluid Passing Through the Porous Medium 269
6.2.3.7.4. Enabling Reactions in a Porous Zone 269
6.2.3.7.5. Including the Relative Velocity Resistance Formulation 269
6.2.3.7.6. Defining the Viscous and Inertial Resistance Coefficients 270
6.2.3.7.7. Deriving Porous Media Inputs Based on Superficial Velocity, Using a Known Pressure Loss 273
6.2.3.7.8. Using the Ergun Equation to Derive Porous Media Inputs for a Packed Bed 274

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.2.3.7.9. Using an Empirical Equation to Derive Porous Media Inputs for Turbulent Flow Through a Perforated Plate 274
6.2.3.7.10. Using Tabulated Data to Derive Porous Media Inputs for Laminar Flow Through a Fibrous Mat 275
6.2.3.7.11. Deriving the Porous Coefficients Based on Experimental Pressure and Velocity Data 275
6.2.3.7.12. Using the Power-Law Model 276
6.2.3.7.13. Defining Porosity 277
6.2.3.7.14. Specifying the Heat Transfer Settings 277
6.2.3.7.14.1. Equilibrium Thermal Model 277
6.2.3.7.14.2. Non-Equilibrium Thermal Model 277
6.2.3.7.15. Specifying the Relative Viscosity 280
6.2.3.7.16. Specifying the Relative Permeability 280
6.2.3.7.17. Defining Sources 283
6.2.3.7.18. Defining Fixed Values 283
6.2.3.7.19. Suppressing the Turbulent Viscosity in the Porous Region 283
6.2.3.7.20. Specifying the Rotation Axis and Defining Zone Motion 283
6.2.3.8. Modeling Porous Media Based on Physical Velocity 283
6.2.3.8.1. Single Phase Porous Media 284
6.2.3.8.2. Multiphase Porous Media 284
6.2.3.8.2.1. The Continuity Equation 285
6.2.3.8.2.2. The Momentum Equation 285
6.2.3.8.2.3. The Energy Equation 285
6.2.3.9. Solution Strategies for Porous Media 286
6.2.3.10. Postprocessing for Porous Media 286
6.2.4. 3D Fan Zones 287
6.2.4.1. Momentum Equations for 3D Fan Zones 287
6.2.4.2. User Inputs for 3D Fan Zones 289
6.2.4.2.1. Defining the Geometry of a 3D Fan Zone 290
6.2.4.2.2. Defining the Properties of a 3D Fan Zone 291
6.2.4.3. 3D Fan Zone Limitations 292
6.2.5. Fixing the Values of Variables 292
6.2.5.1. Overview of Fixing the Value of a Variable 293
6.2.5.1.1. Variables That Can Be Fixed 294
6.2.5.2. Procedure for Fixing Values of Variables in a Zone 294
6.2.5.2.1. Fixing Velocity Components 295
6.2.5.2.2. Fixing Temperature and Enthalpy 295
6.2.5.2.3. Fixing Species Mass Fractions 296
6.2.5.2.4. Fixing Turbulence Quantities 296
6.2.5.2.5. Fixing User-Defined Scalars 296
6.2.6. Locking the Temperature for Solid and Shell Zones 297
6.2.7. Defining Mass, Momentum, Energy, and Other Sources 297
6.2.7.1. Sign Conventions and Units 298
6.2.7.2. Procedure for Defining Sources 298
6.2.7.2.1. Mass Sources 299
6.2.7.2.2. Momentum Sources 299
6.2.7.2.3. Energy Sources 299
6.2.7.2.4. Turbulence Sources 299
6.2.7.2.4.1. Turbulence Sources for the k-ε Model 299
6.2.7.2.4.2. Turbulence Sources for the Spalart-Allmaras Model 300
6.2.7.2.4.3. Turbulence Sources for the k-ω Model 300
6.2.7.2.4.4. Turbulence Sources for the Reynolds Stress Model 300

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.2.7.2.5. Mean Mixture Fraction and Variance Sources 300
6.2.7.2.6. P-1 Radiation Sources 300
6.2.7.2.7. Progress Variable Sources 301
6.2.7.2.8. NO, HCN, and NH3 Sources for the NOx Model 301
6.2.7.2.9. User-Defined Scalar (UDS) Sources 301
6.3. Boundary Conditions 301
6.3.1. Flow Inlet and Exit Boundary Conditions 302
6.3.2. Using Flow Boundary Conditions 302
6.3.2.1. Determining Turbulence Parameters 303
6.3.2.1.1. Specification of Turbulence Quantities Using Profiles 303
6.3.2.1.2. Uniform Specification of Turbulence Quantities 304
6.3.2.1.3. Turbulence Intensity 304
6.3.2.1.4. Turbulence Length Scale and Hydraulic Diameter 304
6.3.2.1.5. Turbulent Viscosity Ratio 305
6.3.2.1.6. Relationships for Deriving Turbulence Quantities 305
6.3.2.1.7. Estimating Modified Turbulent Viscosity from Turbulence Intensity and Length Scale 306
6.3.2.1.8. Estimating Turbulent Kinetic Energy from Turbulence Intensity 306
6.3.2.1.9. Estimating Turbulent Dissipation Rate from a Length Scale 306
6.3.2.1.10. Estimating Turbulent Dissipation Rate from Turbulent Viscosity Ratio 306
6.3.2.1.11. Estimating Turbulent Dissipation Rate for Decaying Turbulence 307
6.3.2.1.12. Estimating Specific Dissipation Rate from a Length Scale 307
6.3.2.1.13. Estimating Specific Dissipation Rate from Turbulent Viscosity Ratio 307
6.3.2.1.14. Estimating Reynolds Stress Components from Turbulent Kinetic Energy 307
6.3.2.1.15. Specifying Inlet Turbulence for LES 308
6.3.3. Pressure Inlet Boundary Conditions 308
6.3.3.1. Inputs at Pressure Inlet Boundaries 308
6.3.3.1.1. Summary 308
6.3.3.1.1.1. Pressure Inputs and Hydrostatic Head 309
6.3.3.1.1.2. Defining Total Pressure and Temperature 310
6.3.3.1.1.3. Defining the Flow Direction 311
6.3.3.1.1.4. Defining Static Pressure 314
6.3.3.1.1.5. Defining Turbulence Parameters 314
6.3.3.1.1.6. Defining Radiation Parameters 314
6.3.3.1.1.7. Defining Species Mass or Mole Fractions 314
6.3.3.1.1.8. Defining Non-Premixed Combustion Parameters 314
6.3.3.1.1.9. Defining Premixed Combustion Boundary Conditions 314
6.3.3.1.1.10. Defining Discrete Phase Boundary Conditions 315
6.3.3.1.1.11. Defining Multiphase Boundary Conditions 315
6.3.3.1.1.12. Defining Open Channel Boundary Conditions 315
6.3.3.2. Default Settings at Pressure Inlet Boundaries 315
6.3.3.3. Calculation Procedure at Pressure Inlet Boundaries 315
6.3.3.3.1. Incompressible Flow Calculations at Pressure Inlet Boundaries 315
6.3.3.3.2. Compressible Flow Calculations at Pressure Inlet Boundaries 316
6.3.4. Velocity Inlet Boundary Conditions 316
6.3.4.1. Inputs at Velocity Inlet Boundaries 317
6.3.4.1.1. Summary 317
6.3.4.1.2. Defining the Velocity 318
6.3.4.1.3. Setting the Velocity Magnitude and Direction 319
6.3.4.1.4. Setting the Velocity Magnitude Normal to the Boundary 319
6.3.4.1.5. Setting the Velocity Components 319
6.3.4.1.6. Setting the Angular Velocity 320

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.4.1.7. Defining Static Pressure 320
6.3.4.1.8. Defining the Temperature 320
6.3.4.1.9. Defining Outflow Gauge Pressure 320
6.3.4.1.10. Defining Turbulence Parameters 320
6.3.4.1.11. Defining Radiation Parameters 320
6.3.4.1.12. Defining Species Mass or Mole Fractions 321
6.3.4.1.13. Defining Non-Premixed Combustion Parameters 321
6.3.4.1.14. Defining Premixed Combustion Boundary Conditions 321
6.3.4.1.15. Defining Discrete Phase Boundary Conditions 321
6.3.4.1.16. Defining Multiphase Boundary Conditions 321
6.3.4.2. Default Settings at Velocity Inlet Boundaries 321
6.3.4.3. Calculation Procedure at Velocity Inlet Boundaries 322
6.3.4.3.1. Treatment of Velocity Inlet Conditions at Flow Inlets 322
6.3.4.3.2. Treatment of Velocity Inlet Conditions at Flow Exits 322
6.3.4.3.3. Density Calculation 322
6.3.5. Mass-Flow Inlet Boundary Conditions 322
6.3.5.1. Limitations and Special Considerations 323
6.3.5.2. Inputs at Mass-Flow Inlet Boundaries 323
6.3.5.2.1. Summary 323
6.3.5.2.2. Selecting the Reference Frame 324
6.3.5.2.3. Defining the Mass Flow Rate or Mass Flux 324
6.3.5.2.4. More About Mass Flux and Average Mass Flux 325
6.3.5.2.5. Defining the Total Temperature 326
6.3.5.2.6. Defining Static Pressure 326
6.3.5.2.7. Defining the Flow Direction 327
6.3.5.2.8. Defining Turbulence Parameters 328
6.3.5.2.9. Defining Radiation Parameters 328
6.3.5.2.10. Defining Species Mass or Mole Fractions 328
6.3.5.2.11. Defining Non-Premixed Combustion Parameters 328
6.3.5.2.12. Defining Premixed Combustion Boundary Conditions 328
6.3.5.2.13. Defining Discrete Phase Boundary Conditions 328
6.3.5.2.14. Defining Open Channel Boundary Conditions 328
6.3.5.3. Default Settings at Mass-Flow Inlet Boundaries 329
6.3.5.4. Calculation Procedure at Mass-Flow Inlet Boundaries 329
6.3.5.4.1. Flow Calculations at Mass Flow Boundaries for Ideal Gases 329
6.3.5.4.2. Flow Calculations at Mass Flow Boundaries for Incompressible Flows 330
6.3.5.4.3. Flux Calculations at Mass Flow Boundaries 330
6.3.6. Mass-Flow Outlet Boundary Conditions 330
6.3.6.1. Limitations 330
6.3.6.2. Inputs at Mass-Flow Outlet Boundaries 330
6.3.6.2.1. Summary 330
6.3.6.2.2. Selecting the Reference Frame 331
6.3.6.2.3. Defining the Mass Flow Rate or Mass Flux 331
6.3.6.2.4. More About Mass Flux and Average Mass Flux 332
6.3.6.2.5. Defining the Flow Direction 332
6.3.6.2.6. Defining Radiation Parameters 333
6.3.6.2.7. Defining Discrete Phase Boundary Conditions 333
6.3.6.3. Default Settings at Mass-Flow Outlet Boundaries 333
6.3.6.4. Calculation Procedure at Mass-Flow Outlet Boundaries 334
6.3.7. Inlet Vent Boundary Conditions 334
6.3.7.1. Inputs at Inlet Vent Boundaries 334
6.3.7.1.1. Specifying the Loss Coefficient 335

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.8. Intake Fan Boundary Conditions 336
6.3.8.1. Inputs at Intake Fan Boundaries 336
6.3.8.1.1. Specifying the Pressure Jump 338
6.3.9. Pressure Outlet Boundary Conditions 338
6.3.9.1. Inputs at Pressure Outlet Boundaries 338
6.3.9.1.1. Summary 338
6.3.9.1.2. Defining Static Pressure 339
6.3.9.1.3. Defining Backflow Conditions 340
6.3.9.1.4. Defining Radiation Parameters 342
6.3.9.1.5. Defining Discrete Phase Boundary Conditions 342
6.3.9.1.6. Defining Open Channel Boundary Conditions 342
6.3.9.2. Default Settings at Pressure Outlet Boundaries 342
6.3.9.3. Calculation Procedure at Pressure Outlet Boundaries 342
6.3.9.3.1. Average Pressure Specification 342
6.3.9.3.1.1. Strong Averaging 343
6.3.9.3.1.2. Weak Averaging 343
6.3.9.4. Other Optional Inputs at Pressure Outlet Boundaries 344
6.3.9.4.1. Non-Reflecting Boundary Conditions Option 344
6.3.9.4.2. Target Mass Flow Rate Option 344
6.3.10. Pressure Far-Field Boundary Conditions 347
6.3.10.1. Limitations 347
6.3.10.2. Inputs at Pressure Far-Field Boundaries 347
6.3.10.2.1. Summary 347
6.3.10.2.2. Defining Static Pressure, Mach Number, and Static Temperature 348
6.3.10.2.3. Defining the Flow Direction 348
6.3.10.2.4. Defining Turbulence Parameters 349
6.3.10.2.5. Defining Radiation Parameters 349
6.3.10.2.6. Defining Species Transport Parameters 349
6.3.10.3. Defining Discrete Phase Boundary Conditions 349
6.3.10.4. Default Settings at Pressure Far-Field Boundaries 349
6.3.10.5. Calculation Procedure at Pressure Far-Field Boundaries 349
6.3.11. Outflow Boundary Conditions 350
6.3.11.1. ANSYS Fluent's Treatment at Outflow Boundaries 351
6.3.11.2. Using Outflow Boundaries 351
6.3.11.3. Mass Flow Split Boundary Conditions 352
6.3.11.4. Other Inputs at Outflow Boundaries 353
6.3.11.4.1. Radiation Inputs at Outflow Boundaries 353
6.3.11.4.2. Defining Discrete Phase Boundary Conditions 353
6.3.12. Outlet Vent Boundary Conditions 353
6.3.12.1. Inputs at Outlet Vent Boundaries 353
6.3.12.1.1. Specifying the Loss Coefficient 355
6.3.13. Exhaust Fan Boundary Conditions 355
6.3.13.1. Inputs at Exhaust Fan Boundaries 355
6.3.13.1.1. Specifying the Pressure Jump 356
6.3.14. Degassing Boundary Conditions 357
6.3.14.1. Limitations 358
6.3.14.2. Inputs at Degassing Boundaries 358
6.3.15. Wall Boundary Conditions 358
6.3.15.1. Inputs at Wall Boundaries 358
6.3.15.1.1. Summary 358
6.3.15.2. Wall Motion 359
6.3.15.2.1. Defining a Stationary Wall 359

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.15.2.2. Velocity Conditions for Moving Walls 359
6.3.15.2.3. Shear Conditions at Walls 361
6.3.15.2.4. No-Slip Walls 362
6.3.15.2.5. Specified Shear 362
6.3.15.2.6. Specularity Coefficient 362
6.3.15.2.7. Marangoni Stress 363
6.3.15.2.8. Wall Roughness Effects in Turbulent Wall-Bounded Flows 364
6.3.15.2.8.1. Standard Law-of-the-Wall Modified for Roughness 364
6.3.15.2.8.1.1. Setting the Roughness Parameters 368
6.3.15.2.8.2. Additional Roughness Models for Icing Simulations 368
6.3.15.2.8.2.1. Specified Roughness 369
6.3.15.2.8.2.2. NASA Correlation 370
6.3.15.2.8.2.3. Shin-et-al 370
6.3.15.2.8.2.4. ICE3D Roughness File 370
6.3.15.3. Thermal Boundary Conditions at Walls 370
6.3.15.3.1. Heat Flux Boundary Conditions 372
6.3.15.3.2. Temperature Boundary Conditions 372
6.3.15.3.3. Convective Heat Transfer Boundary Conditions 372
6.3.15.3.4. External Radiation Boundary Conditions 372
6.3.15.3.5. Combined Convection and External Radiation Boundary Conditions 373
6.3.15.3.6. Augmented Heat Transfer 373
6.3.15.3.7. Thin-Wall Thermal Resistance Parameters 373
6.3.15.3.8. Thermal Conditions for Two-Sided Walls 375
6.3.15.3.9. Shell Conduction 376
6.3.15.3.10. Heat Transfer Boundary Conditions Through System Coupling 377
6.3.15.3.11. Heat Transfer Boundary Conditions Across a Mapped Interface 378
6.3.15.4. Species Boundary Conditions for Walls 379
6.3.15.4.1. Reaction Boundary Conditions for Walls 380
6.3.15.5. Radiation Boundary Conditions for Walls 381
6.3.15.6. Discrete Phase Model (DPM) Boundary Conditions for Walls 381
6.3.15.6.1. Wall Adhesion Contact Angle for VOF Model 381
6.3.15.7. User-Defined Scalar (UDS) Boundary Conditions for Walls 381
6.3.15.8. Wall Film Conditions for Walls 381
6.3.15.9. Default Settings at Wall Boundaries 381
6.3.15.10. Shear-Stress Calculation Procedure at Wall Boundaries 381
6.3.15.10.1. Shear-Stress Calculation in Laminar Flow 382
6.3.15.10.2. Shear-Stress Calculation in Turbulent Flows 382
6.3.15.11. Heat Transfer Calculations at Wall Boundaries 382
6.3.15.11.1. Temperature Boundary Conditions 382
6.3.15.11.2. Heat Flux Boundary Conditions 383
6.3.15.11.3. Convective Heat Transfer Boundary Conditions 383
6.3.15.11.4. External Radiation Boundary Conditions 383
6.3.15.11.5. Combined External Convection and Radiation Boundary Conditions 384
6.3.15.11.6. Calculation of the Fluid-Side Heat Transfer Coefficient 384
6.3.16. Symmetry Boundary Conditions 384
6.3.16.1. Examples of Symmetry Boundaries 384
6.3.16.2. Calculation Procedure at Symmetry Boundaries 386
6.3.17. Periodic Boundary Conditions 386
6.3.17.1. Examples of Periodic Boundaries 387
6.3.17.2. Inputs for Periodic Boundaries 387
6.3.17.3. Default Settings at Periodic Boundaries 389
6.3.17.4. Calculation Procedure at Periodic Boundaries 389

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.18. Axis Boundary Conditions 389
6.3.18.1. Calculation Procedure at Axis Boundaries 389
6.3.19. Fan Boundary Conditions 389
6.3.19.1. Fan Equations 390
6.3.19.1.1. Modeling the Pressure Rise Across the Fan 390
6.3.19.1.2. Modeling the Fan Swirl Velocity 390
6.3.19.2. User Inputs for Fans 391
6.3.19.2.1. Identifying the Fan Zone 391
6.3.19.2.2. Defining the Pressure Jump 392
6.3.19.2.2.1. Polynomial, Piecewise-Linear, or Piecewise-Polynomial Function 392
6.3.19.2.2.2. Constant Value 393
6.3.19.2.2.3. User-Defined Function or Profile 393
6.3.19.2.2.4. Example: Determining the Pressure Jump Function 393
6.3.19.2.3. Defining Discrete Phase Boundary Conditions for the Fan 394
6.3.19.2.4. Defining the Fan Swirl Velocity 394
6.3.19.2.4.1. Polynomial Function 395
6.3.19.2.4.2. Constant Value 395
6.3.19.2.4.3. User-Defined Function or Profile 395
6.3.19.3. Postprocessing for Fans 396
6.3.19.3.1. Reporting the Pressure Rise Through the Fan 396
6.3.19.3.2. Graphical Plots 396
6.3.20. Radiator Boundary Conditions 396
6.3.20.1. Radiator Equations 396
6.3.20.1.1. Modeling the Pressure Loss Through a Radiator 396
6.3.20.1.2. Modeling the Heat Transfer Through a Radiator 397
6.3.20.1.2.1. Calculating the Heat Transfer Coefficient 397
6.3.20.2. User Inputs for Radiators 398
6.3.20.2.1. Identifying the Radiator Zone 399
6.3.20.2.2. Defining the Pressure Loss Coefficient Function 399
6.3.20.2.2.1. Polynomial, Piecewise-Linear, or Piecewise-Polynomial Function 399
6.3.20.2.2.2. Constant Value 400
6.3.20.2.2.3. Example: Calculating the Loss Coefficient 400
6.3.20.2.3. Defining the Heat Flux Parameters 401
6.3.20.2.3.1. Polynomial, Piecewise-Linear, or Piecewise-Polynomial Function 401
6.3.20.2.3.2. Constant Value 401
6.3.20.2.3.3. Example: Determining the Heat Transfer Coefficient Function 401
6.3.20.2.4. Defining Discrete Phase Boundary Conditions for the Radiator 402
6.3.20.3. Postprocessing for Radiators 402
6.3.20.3.1. Reporting the Radiator Pressure Drop 402
6.3.20.3.2. Reporting Heat Transfer in the Radiator 402
6.3.20.3.3. Graphical Plots 402
6.3.21. Porous Jump Boundary Conditions 403
6.3.21.1. User Inputs for the Porous Jump Model 404
6.3.21.1.1. Identifying the Porous Jump Zone 405
6.3.21.1.2. Defining Discrete Phase Boundary Conditions for the Porous Jump 405
6.3.21.2. Postprocessing for the Porous Jump 405
6.4. Boundary Acoustic Wave Models 405
6.4.1. Turbo-Specific Non-Reflecting Boundary Conditions 406
6.4.1.1. Overview 406
6.4.1.2. Limitations 406
6.4.1.3. Theory 409
6.4.1.3.1. Equations in Characteristic Variable Form 409

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.4.1.3.2. Inlet Boundary 411
6.4.1.3.3. Outlet Boundary 413
6.4.1.3.4. Updated Flow Variables 414
6.4.1.4. Using Turbo-Specific Non-Reflecting Boundary Conditions 414
6.4.1.4.1. Using the NRBCs with the Mixing-Plane Model 415
6.4.1.4.2. Using the NRBCs in Parallel ANSYS Fluent 415
6.4.2. General Non-Reflecting Boundary Conditions 416
6.4.2.1. Overview 416
6.4.2.2. Restrictions and Limitations 416
6.4.2.3. Theory 416
6.4.2.4. Using the General Non-Reflecting Boundary Condition 421
6.4.3. Impedance Boundary Conditions 422
6.4.3.1. Restrictions and Limitations 423
6.4.3.2. Theory 423
6.4.3.3. Using the Impedance Boundary Condition 424
6.4.4. Transparent Flow Forcing Boundary Conditions 425
6.4.4.1. Restrictions and Limitations 425
6.4.4.2. Theory 425
6.4.4.3. Using the Transparent Flow Forcing Boundary Condition 426
6.5. User-Defined Fan Model 427
6.5.1. Steps for Using the User-Defined Fan Model 428
6.5.2. Example of a User-Defined Fan 428
6.5.2.1. Setting the User-Defined Fan Parameters 429
6.5.2.2. Sample User-Defined Fan Program 430
6.5.2.3. Initializing the Flow Field and Profile Files 432
6.5.2.4. Selecting the Profiles 432
6.5.2.5. Performing the Calculation 433
6.5.2.6. Results 434
6.6. Profiles 435
6.6.1. Profile Specification Types 435
6.6.2. Profile File Formats 436
6.6.2.1. Standard Profiles 436
6.6.2.1.1. Example 437
6.6.2.2. CSV Profiles 438
6.6.3. Using Profiles 440
6.6.3.1. Checking and Deleting Profiles 441
6.6.3.2. Viewing Profile Data 442
6.6.3.3. Example 443
6.6.4. Reorienting Profiles 443
6.6.4.1. Steps for Changing the Profile Orientation 444
6.6.4.2. Profile Orienting Example 446
6.6.5. Defining Transient Cell Zone and Boundary Conditions 448
6.6.5.1. Standard Transient Profiles 449
6.6.5.2. Tabular Transient Profiles 450
6.6.5.3. Profiles for Moving and Deforming Meshes 451
6.7. Coupling Boundary Conditions with GT-POWER 451
6.7.1. Requirements and Restrictions 451
6.7.2. User Inputs 453
6.7.3. Torque-Speed Coupling with GT-POWER 455
6.8. Coupling Boundary Conditions with WAVE 456
6.8.1. Requirements and Restrictions 456
6.8.2. User Inputs 457

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Physical Properties 459
7.1. Defining Materials 459
7.1.1. Physical Properties for Solid Materials 460
7.1.2. Material Types and Databases 460
7.1.3. Using the **Create/Edit Materials** Dialog Box 461
7.1.3.1. Modifying Properties of an Existing Material 463
7.1.3.2. Renaming an Existing Material 463
7.1.3.3. Copying Materials from the ANSYS Fluent Database 464
7.1.3.4. Creating a New Material 466
7.1.3.5. Saving Materials and Properties 466
7.1.3.6. Deleting a Material 466
7.1.3.7. Changing the Order of the Materials List 467
7.1.4. Using a User-Defined Materials Database 467
7.1.4.1. Opening a User-Defined Database 468
7.1.4.2. Viewing Materials in a User-Defined Database 468
7.1.4.3. Copying Materials from a User-Defined Database 469
7.1.4.4. Copying Materials from the Case to a User-Defined Database 470
7.1.4.5. Modifying Properties of an Existing Material 471
7.1.4.6. Creating a New Materials Database and Materials 472
7.1.4.7. Deleting Materials from a Database 474
7.2. Defining Properties Using Temperature-Dependent Functions 475
7.2.1. Inputs for Polynomial Functions 475
7.2.2. Inputs for Piecewise-Linear Functions 476
7.2.3. Inputs for Piecewise-Polynomial Functions 478
7.2.4. Checking and Modifying Existing Profiles 479
7.3. Density 479
7.3.1. Defining Density for Various Flow Regimes 479
7.3.1.1. Mixing Density Relationships in Multiple-Zone Models 480
7.3.2. Input of Constant Density 480
7.3.3. Inputs for the Boussinesq Approximation 480
7.3.4. Compressible Liquid Density Method 480
7.3.4.1. Compressible Liquid Inputs 481
7.3.4.2. Compressible Liquid Density Method Availability 484
7.3.5. Density as a Profile Function of Temperature 484
7.3.6. Incompressible Ideal Gas Law 484
7.3.6.1. Density Inputs for the Incompressible Ideal Gas Law 484
7.3.7. Ideal Gas Law for Compressible Flows 485
7.3.7.1. Density Inputs for the Ideal Gas Law for Compressible Flows 485
7.3.8. Composition-Dependent Density for Multicomponent Mixtures 486
7.4. Viscosity 487
7.4.1. Input of Constant Viscosity 488
7.4.2. Viscosity as a Function of Temperature 488
7.4.2.1. Sutherland Viscosity Law 489
7.4.2.1.1. Inputs for Sutherland's Law 489
7.4.2.2. Power-Law Viscosity Law 489
7.4.2.2.1. Inputs for the Power Law 490
7.4.3. Defining the Viscosity Using Kinetic Theory 490
7.4.4. Composition-Dependent Viscosity for Multicomponent Mixtures 491
7.4.5. Viscosity for Non-Newtonian Fluids 492
7.4.5.1. Temperature Dependent Viscosity 492
7.4.5.2. Power Law for Non-Newtonian Viscosity 493
7.4.5.2.1. Inputs for the Non-Newtonian Power Law 493

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.5.3. The Carreau Model for Pseudo-Plastics 494
7.4.5.3.1. Inputs for the Carreau Model 494
7.4.5.4. Cross Model 495
7.4.5.4.1. Inputs for the Cross Model 495
7.4.5.5. Herschel-Bulkley Model for Bingham Plastics 495
7.4.5.5.1. Inputs for the Herschel-Bulkley Model 496
7.5. Thermal Conductivity 497
7.5.1. Constant Thermal Conductivity 498
7.5.2. Thermal Conductivity as a Function of Temperature 498
7.5.3. Thermal Conductivity Using Kinetic Theory 499
7.5.4. Composition-Dependent Thermal Conductivity for Multicomponent Mixtures 499
7.5.5. Anisotropic Thermal Conductivity for Solids 500
7.5.5.1. Anisotropic Thermal Conductivity 501
7.5.5.2. Biaxial Thermal Conductivity 502
7.5.5.3. Orthotropic Thermal Conductivity 503
7.5.5.4. Cylindrical Orthotropic Thermal Conductivity 504
7.5.5.5. Principal Axes and Principal Values 506
7.5.5.6. User-Defined Anisotropic Thermal Conductivity 507
7.6. User-Defined Scalar (UDS) Diffusivity 508
7.6.1. Isotropic Diffusion 508
7.6.2. Anisotropic Diffusion 509
7.6.2.1. Anisotropic Diffusivity 510
7.6.2.2. Orthotropic Diffusivity 511
7.6.2.3. Cylindrical Orthotropic Diffusivity 512
7.6.3. User-Defined Anisotropic Diffusivity 513
7.7. Specific Heat Capacity 513
7.7.1. Input of Constant Specific Heat Capacity 514
7.7.2. Specific Heat Capacity as a Function of Temperature 514
7.7.3. Defining Specific Heat Capacity Using Kinetic Theory 515
7.7.4. Specific Heat Capacity as a Function of Composition 515
7.8. Radiation Properties 515
7.8.1. Absorption Coefficient 516
7.8.1.1. Inputs for a Constant Absorption Coefficient 516
7.8.1.2. Inputs for a Composition-Dependent Absorption Coefficient 516
7.8.1.2.1. Path Length Inputs 516
7.8.1.2.1.1. Inputs for a Non-Gray Radiation Absorption Coefficient 517
7.8.1.2.1.2. Effect of Particles and Soot on the Absorption Coefficient 517
7.8.2. Scattering Coefficient 517
7.8.2.1. Inputs for a Constant Scattering Coefficient 518
7.8.2.2. Inputs for the Scattering Phase Function 518
7.8.2.2.1. Isotropic Phase Function 518
7.8.2.2.2. Linear-Anisotropic Phase Function 518
7.8.2.2.3. Delta-Eddington Phase Function 518
7.8.2.2.4. User-Defined Phase Function 518
7.8.3. Refractive Index 518
7.8.4. Reporting the Radiation Properties 519
7.9. Mass Diffusion Coefficients 519
7.9.1. Fickian Diffusion 519
7.9.2. Full Multicomponent Diffusion 520
7.9.2.1. General Theory 520
7.9.2.2. Maxwell-Stefan Equations 520
7.9.3. Anisotropic Species Diffusion 521

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.9.4. Thermal Diffusion Coefficients 522
7.9.4.1. Thermal Diffusion Coefficient Inputs 523
7.9.5. Mass Diffusion Coefficient Inputs 524
7.9.5.1. Constant Dilute Approximation Inputs 524
7.9.5.2. Dilute Approximation Inputs 525
7.9.5.3. Multicomponent Method Inputs 526
7.9.5.4. Unity Lewis Number 527
7.9.6. Mass Diffusion Coefficient Inputs for Turbulent Flow 527
7.10. Standard State Enthalpies 528
7.11. Standard State Entropies 528
7.12. Unburnt Thermal Diffusivity 529
7.13. Kinetic Theory Parameters 529
7.13.1. Inputs for Kinetic Theory 529
7.14. Operating Pressure 530
7.14.1. The Significance of Operating Pressure 530
7.14.2. Operating Pressure, Gauge Pressure, and Absolute Pressure 531
7.14.3. Setting the Operating Pressure 531
7.15. Reference Pressure Location 531
7.15.1. Actual Reference Pressure Location 532
7.16. Real Gas Models 532
7.16.1. Introduction 533
7.16.2. Choosing a Real Gas Model 534
7.16.3. Cubic Equation of State Models 535
7.16.3.1. Overview and Limitations 535
7.16.3.2. Equation of State 537
7.16.3.3. Enthalpy, Entropy, and Specific Heat Calculations 538
7.16.3.4. Critical Constants for Pure Components 540
7.16.3.5. Calculations for Mixtures 541
7.16.3.5.1. Using the Cubic Equation of State Real Gas Models 542
7.16.3.5.2. Solution Strategies and Considerations for Cubic Equations of State Real Gas Models 546
7.16.3.5.3. Using the Cubic Equation of State Models with the Lagrangian Dispersed Phase Models 548
7.16.3.5.4. Postprocessing the Cubic Equations of State Real Gas Model 550
7.16.4. The NIST Real Gas Models 550
7.16.4.1. Limitations of the NIST Real Gas Models 551
7.16.4.2. The REFPROP v9.1 Database 551
7.16.4.3. Using the NIST Real Gas Models 553
7.16.4.3.1. Activating the NIST Real Gas Model 554
7.16.4.3.2. Creating Full NIST Look-up Tables 555
7.16.4.3.3. Creating Binary Mixture Saturation Tables for Binary Mixtures 558
7.16.4.3.4. Changing the REFPROP Library and Fluid Files 560
7.16.4.4. Solution Strategies and Considerations for NIST Real Gas Model Simulation 560
7.16.4.4.1. Writing Your Case File 561
7.16.4.4.2. Postprocessing 561
7.16.5. The User-Defined Real Gas Model 561
7.16.5.1. Limitations of the User-Defined Real Gas Model 561
7.16.5.2. Writing the UDRGM C Function Library 564
7.16.5.3. Compiling Your UDRGM C Functions and Building a Shared Library File 567
7.16.5.3.1. Compiling the UDRGM Using the Graphical Interface 567
7.16.5.3.2. Compiling the UDRGM Using the Text Interface 567
7.16.5.3.3. Loading the UDRGM Shared Library File 568

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.16.5.4. UDRGM Example: Ideal Gas Equation of State 569
7.16.5.4.1. Ideal Gas UDRGM Code Listing 570
7.16.5.5. Additional UDRGM Examples 572
8. Modeling Basic Fluid Flow 573
8.1. User-Defined Scalar (UDS) Transport Equations 573
8.1.1. Introduction 573
8.1.2. UDS Theory 573
8.1.2.1. Single Phase Flow 574
8.1.2.2. Multiphase Flow 574
8.1.3. Setting Up UDS Equations in ANSYS Fluent 575
8.1.3.1. Single Phase Flow 576
8.1.3.2. Multiphase Flow 581
8.2. Periodic Flows 582
8.2.1. Overview and Limitations 582
8.2.1.1. Overview 583
8.2.1.2. Limitations for Modeling Streamwise-Periodic Flow 583
8.2.2. User Inputs for the Pressure-Based Solver 584
8.2.2.1. Setting Parameters for the Calculation of β 585
8.2.3. User Inputs for the Density-Based Solvers 585
8.2.4. Monitoring the Value of the Pressure Gradient 586
8.2.5. Postprocessing for Streamwise-Periodic Flows 586
8.3. Swirling and Rotating Flows 587
8.3.1. Overview of Swirling and Rotating Flows 588
8.3.1.1. Axisymmetric Flows with Swirl or Rotation 588
8.3.1.1.1. Momentum Conservation Equation for Swirl Velocity 588
8.3.1.2. Three-Dimensional Swirling Flows 588
8.3.1.3. Flows Requiring a Moving Reference Frame 588
8.3.2. Turbulence Modeling in Swirling Flows 589
8.3.3. Mesh Setup for Swirling and Rotating Flows 589
8.3.3.1. Coordinate System Restrictions 589
8.3.3.2. Mesh Sensitivity in Swirling and Rotating Flows 589
8.3.4. Modeling Axisymmetric Flows with Swirl or Rotation 589
8.3.4.1. Problem Setup for Axisymmetric Swirling Flows 590
8.3.4.2. Solution Strategies for Axisymmetric Swirling Flows 591
8.3.4.2.1. Step-By-Step Solution Procedures for Axisymmetric Swirling Flows 591
8.3.4.2.2. Improving Solution Stability by Gradually Increasing the Rotational or Swirl Speed 592
8.3.4.2.2.1. Postprocessing for Axisymmetric Swirling Flows 593
8.4. Compressible Flows 593
8.4.1. When to Use the Compressible Flow Model 594
8.4.2. Physics of Compressible Flows 595
8.4.2.1. Basic Equations for Compressible Flows 595
8.4.2.2. The Compressible Form of the Gas Law 595
8.4.3. Modeling Inputs for Compressible Flows 596
8.4.3.1. Boundary Conditions for Compressible Flows 597
8.4.4. Floating Operating Pressure 597
8.4.4.1. Limitations 597
8.4.4.2. Theory 597
8.4.4.3. Enabling Floating Operating Pressure 598
8.4.4.4. Setting the Initial Value for the Floating Operating Pressure 598
8.4.4.5. Storage and Reporting of the Floating Operating Pressure 598
8.4.4.6. Monitoring Absolute Pressure 598

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.4.5. Solution Strategies for Compressible Flows 599
8.4.6. Reporting of Results for Compressible Flows 599
8.5. Inviscid Flows 599
8.5.1. Setting Up an Inviscid Flow Model 600
8.5.2. Solution Strategies for Inviscid Flows 600
8.5.3. Postprocessing for Inviscid Flows 601
9. Modeling Flows with Moving Reference Frames 603
9.1. Introduction 603
9.2. Flow in Single Moving Reference Frames (SRF) 605
9.2.1. Mesh Setup for a Single Moving Reference Frame 605
9.2.2. Setting Up a Single Moving Reference Frame Problem 606
9.2.2.1. Choosing the Relative or Absolute Velocity Formulation 609
9.2.2.1.1. Example 609
9.2.3. Solution Strategies for a Single Moving Reference Frame 610
9.2.3.1. Gradual Increase of the Rotational Speed to Improve Solution Stability 611
9.2.4. Postprocessing for a Single Moving Reference Frame 611
9.3. Flow in Multiple Moving Reference Frames 612
9.3.1. The Multiple Reference Frame Model 613
9.3.1.1. Overview 613
9.3.1.2. Limitations 614
9.3.2. The Mixing Plane Model 615
9.3.2.1. Overview 615
9.3.2.2. Limitations 615
9.3.3. Mesh Setup for a Multiple Moving Reference Frame 616
9.3.4. Setting Up a Multiple Moving Reference Frame Problem 616
9.3.4.1. Setting Up Multiple Reference Frames 616
9.3.4.2. Setting Up the Mixing Plane Model 619
9.3.4.2.1. Modeling Options 622
9.3.4.2.1.1. Fixing the Pressure Level for an Incompressible Flow 622
9.3.4.2.1.2. Conserving Swirl Across the Mixing Plane 623
9.3.4.2.1.3. Conserving Total Enthalpy Across the Mixing Plane 623
9.3.5. Solution Strategies for MRF and Mixing Plane Problems 624
9.3.5.1. MRF Model 624
9.3.5.2. Mixing Plane Model 624
9.3.6. Postprocessing for MRF and Mixing Plane Problems 624
10. Modeling Flows Using Sliding and Dynamic Meshes 627
10.1. Introduction 627
10.2. Sliding Mesh Examples 628
10.3. The Sliding Mesh Technique 630
10.4. Sliding Mesh Interface Shapes 631
10.5. Using Sliding Meshes 633
10.5.1. Requirements, Constraints, and Considerations 633
10.5.2. Setting Up the Sliding Mesh Problem 634
10.5.3. Solution Strategies for Sliding Meshes 637
10.5.3.1. Saving Case and Data Files 637
10.5.3.2. Time-Periodic Solutions 638
10.5.4. Postprocessing for Sliding Meshes 639
10.6. Using Dynamic Meshes 641
10.6.1. Setting Dynamic Mesh Modeling Parameters 643
10.6.2. Dynamic Mesh Update Methods 644
10.6.2.1. Smoothing Methods 644
10.6.2.1.1. Spring-Based Smoothing 646

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.6.2.1.1.1. Applicability of the Spring-Based Smoothing Method 649
10.6.2.1.2. Diffusion-Based Smoothing 650
10.6.2.1.2.1. Diffusivity Based on Boundary Distance 653
10.6.2.1.2.2. Diffusivity Based on Cell Volume 655
10.6.2.1.2.3. Applicability of the Diffusion-Based Smoothing Method 655
10.6.2.1.3. Linearly Elastic Solid Based Smoothing Method 656
10.6.2.1.3.1. Applicability of the Linearly Elastic Solid Based Smoothing Method 656
10.6.2.1.4. Smoothing from a Reference Position 657
10.6.2.1.5. Laplacian Smoothing Method 657
10.6.2.1.6. Boundary Layer Smoothing Method 658
10.6.2.2. Dynamic Layering 661
10.6.2.2.1. Applicability of the Dynamic Layering Method 664
10.6.2.3. Remeshing Methods 665
10.6.2.3.1. Local Remeshing Method 668
10.6.2.3.1.1. Local Cell Remeshing Method 669
10.6.2.3.1.2. Local Face Remeshing Method 669
10.6.2.3.1.2.1. Applicability of the Local Face Remeshing Method 669
10.6.2.3.1.3. Local Remeshing Based on Size Functions 670
10.6.2.3.2. Cell Zone Remeshing Method 676
10.6.2.3.2.1. Limitations of the Cell Zone Remeshing Method 676
10.6.2.3.3. Face Region Remeshing Method 676
10.6.2.3.3.1. Face Region Remeshing with Wedge Cells in Prism Layers 678
10.6.2.3.3.2. Applicability of the Face Region Remeshing Method 680
10.6.2.3.4. CutCell Zone Remeshing Method 681
10.6.2.3.4.1. Applicability of the CutCell Zone Remeshing Method 683
10.6.2.3.4.2. Using the CutCell Zone Remeshing Method 683
10.6.2.3.4.3. Applying the CutCell Zone Remeshing Method Manually 684
10.6.2.3.5. 2.5D Surface Remeshing Method 685
10.6.2.3.5.1. Applicability of the 2.5D Surface Remeshing Method 686
10.6.2.3.5.2. Using the 2.5D Model 686
10.6.2.4. Volume Mesh Update Procedure 689
10.6.2.5. Transient Considerations for Remeshing and Layering 689
10.6.3. Feature Detection 690
10.6.3.1. Applicability of Feature Detection 690
10.6.4. In-Cylinder Settings 690
10.6.4.1. Using the In-Cylinder Option 695
10.6.4.1.1. Overview 695
10.6.4.1.2. Defining the Mesh Topology 696
10.6.4.1.3. Defining Motion/Geometry Attributes of Mesh Zones 699
10.6.4.1.4. Defining Valve Opening and Closure 705
10.6.5. Six DOF Solver Settings 705
10.6.5.1. Setting Rigid Body Motion Attributes for the Six DOF Solver 707
10.6.6. Implicit Update Settings 709
10.6.7. Contact Detection Settings 711
10.6.8. Defining Dynamic Mesh Events 713
10.6.8.1. Procedure for Defining Events 713
10.6.8.2. Defining Events for In-Cylinder Applications 716
10.6.8.2.1. Events 716
10.6.8.2.2. Changing the Zone Type 717
10.6.8.2.3. Copying Zone Boundary Conditions 717
10.6.8.2.4. Activating a Cell Zone 717
10.6.8.2.5. Deactivating a Cell Zone 717

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.6.8.2.6. Creating a Sliding Interface 717
10.6.8.2.7. Deleting a Sliding Interface 719
10.6.8.2.8. Changing the Motion Attribute of a Dynamic Zone 719
10.6.8.2.9. Changing the Time Step 719
10.6.8.2.10. Changing the Under-Relaxation Factor 719
10.6.8.2.11. Inserting a Boundary Zone Layer 719
10.6.8.2.12. Removing a Boundary Zone Layer 720
10.6.8.2.13. Inserting an Interior Zone Layer 720
10.6.8.2.14. Removing an Interior Zone Layer 721
10.6.8.2.15. Inserting a Cell Layer 721
10.6.8.2.16. Removing a Cell Layer 721
10.6.8.2.17. Executing a Command 722
10.6.8.2.18. Replacing the Mesh 722
10.6.8.2.19. Resetting Inert EGR 722
10.6.8.2.20. Diesel Unsteady Flamelet Reset 722
10.6.8.3. Exporting and Importing Events 722
10.6.9. Specifying the Motion of Dynamic Zones 722
10.6.9.1. General Procedure 723
10.6.9.1.1. Creating a Dynamic Zone 723
10.6.9.1.2. Modifying a Dynamic Zone 723
10.6.9.1.3. Checking the Center of Gravity 723
10.6.9.1.4. Deleting a Dynamic Zone 723
10.6.9.2. Stationary Zones 723
10.6.9.3. Rigid Body Motion 726
10.6.9.4. Deforming Motion 731
10.6.9.5. User-Defined Motion 735
10.6.9.5.1. Specifying Boundary Layer Deformation Smoothing 737
10.6.9.6. System Coupling Motion 738
10.6.9.7. Solution Stabilization for System Coupling and Six DOF Motion 738
10.6.9.8. Solid-Body Kinematics 739
10.6.10. Previewing the Dynamic Mesh 742
10.6.10.1. Previewing Zone Motion 742
10.6.10.2. Previewing Mesh Motion 743
10.6.11. Steady-State Dynamic Mesh Applications 744
10.6.11.1. An Example of Steady-State Dynamic Mesh Usage 746
11. Modeling Turbulence 749
11.1. Introduction 749
11.2. Choosing a Turbulence Model 751
11.2.1. Reynolds Averaged Navier-Stokes (RANS) Turbulence Models 752
11.2.1.1. Spalart-Allmaras One-Equation Model 752
11.2.1.2. k-ε Models 752
11.2.1.3. k-ω Models 753
11.2.1.4. Reynold Stress Models 753
11.2.1.5. Laminar-Turbulent Transition Models 754
11.2.1.6. Curvature Correction for the Spalart-Allmaras and Two-Equation Models 755
11.2.1.7. Production Limiters for Two-Equation Models 755
11.2.1.8. Model Enhancements 755
11.2.1.9. Wall Treatment for RANS Models 755
11.2.1.10. Grid Resolution for RANS Models 756
11.2.2. Scale-Resolving Simulation (SRS) Models 756
11.2.2.1. Large Eddy Simulation (LES) 757
11.2.2.2. Hybrid RANS-LES Models 757

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.2.2.2.1. Scale-Adaptive Simulation (SAS) 758
11.2.2.2.2. Detached Eddy Simulation (DES) 758
11.2.2.2.3. Shielded Detached Eddy Simulation (SDES) and Stress-Blended Eddy Simulation (SBES) 759
11.2.2.3. Zonal Modeling and Embedded LES (ELES) 760
11.2.3. Grid Resolution SRS Models 760
11.2.3.1. Wall Boundary Layers 760
11.2.3.2. Free Shear Flows 760
11.2.4. Numerics Settings for SRS Models 761
11.2.4.1. Time Discretization 761
11.2.4.2. Spatial Discretization 762
11.2.4.3. Iterative Scheme 762
11.2.4.3.1. Convergence Control 763
11.2.5. Model Hierarchy 763
11.3. Steps in Using a Turbulence Model 764
11.4. Setting Up the Spalart-Allmaras Model 766
11.5. Setting Up the k-ε Model 767
11.5.1. Setting Up the Standard or Realizable k-ε Model 767
11.5.2. Setting Up the RNG k-ε Model 769
11.6. Setting Up the k-ω Model 771
11.6.1. Setting Up the Standard k-ω Model 771
11.6.2. Setting Up the Baseline (BSL) k-ω Model 772
11.6.3. Setting Up the Shear-Stress Transport k-ω Model 773
11.7. Setting Up the Transition k-kl-ω Model 775
11.8. Setting Up the Transition SST Model 775
11.9. Setting Up the Intermittency Transition Model 777
11.10. Setting Up the Reynolds Stress Model 778
11.11. Setting Up Scale-Adaptive Simulation (SAS) Modeling 781
11.12. Setting Up the Detached Eddy Simulation Model 783
11.12.1. Setting Up DES with the Spalart-Allmaras Model 784
11.12.2. Setting Up DES with the Realizable k-ε Model 785
11.12.3. Setting Up DES with the SST k-ω Model 786
11.12.4. Setting Up DES with the BSL k-ω Model 787
11.12.5. Setting Up DES with the Transition SST Model 789
11.13. Setting Up the Large Eddy Simulation Model 792
11.14. Model Constants 793
11.15. Setting Up the Embedded Large Eddy Simulation (ELES) Model 793
11.16. Setup Options for All Turbulence Modeling 797
11.16.1. Including the Viscous Heating Effects 798
11.16.2. Including Turbulence Generation Due to Buoyancy 798
11.16.3. Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models 798
11.16.4. Including the Compressibility Effects Option 799
11.16.5. Including Production Limiters for Two-Equation Models 799
11.16.6. Including the Intermittency Transition Model 800
11.16.7. Vorticity- and Strain/Vorticity-Based Production 800
11.16.8. Delayed Detached Eddy Simulation (DDES) 800
11.16.9. Differential Viscosity Modification 800
11.16.10. Swirl Modification 801
11.16.11. Low-Re Corrections 801
11.16.12. Shear Flow Corrections 801
11.16.13. Turbulence Damping 801

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.16.14. Including Pressure Gradient Effects 802
11.16.15. Including Thermal Effects 802
11.16.16. Including the Wall Reflection Term 802
11.16.17. Solving the k Equation to Obtain Wall Boundary Conditions 802
11.16.18. Quadratic Pressure-Strain Model 802
11.16.19. Stress-Omega and Stress-BSL Models 803
11.16.20. Subgrid-Scale Model 803
11.16.21. Customizing the Turbulent Viscosity 804
11.16.22. Customizing the Turbulent Prandtl and Schmidt Numbers 804
11.16.23. Modeling Turbulence with Non-Newtonian Fluids 804
11.16.24. Including Scale-Adaptive Simulation with ω-Based URANS Models 804
11.16.25. Including Detached Eddy Simulation with the Transition SST Model 805
11.16.26. Including the SDES or SBES Model with BSL, SST, and Transition SST Models 805
11.16.27. Shielding Functions for the BSL / SST / Transition SST Detached Eddy Simulation Model 807
11.17. Defining Turbulence Boundary Conditions 807
11.17.1. Wall Roughness Effects 807
11.17.2. The Spalart-Allmaras Model 807
11.17.3. k-ε Models and k-ω Models 808
11.17.4. Reynolds Stress Model 808
11.17.5. Large Eddy Simulation Model 810
11.18. Providing an Initial Guess for k and ε (or k and ω) 810
11.19. Solution Strategies for Turbulent Flow Simulations 810
11.19.1. Mesh Generation 811
11.19.2. Accuracy 811
11.19.3. Convergence 811
11.19.4. RSM-Specific Solution Strategies 812
11.19.4.1. Under-Relaxation of the Reynolds Stresses 812
11.19.4.2. Disabling Calculation Updates of the Reynolds Stresses 812
11.19.4.3. Residual Reporting for the RSM 813
11.19.5. LES-Specific Solution Strategies 813
11.19.5.1. Temporal Discretization 814
11.19.5.2. Spatial Discretization 814
11.20. Postprocessing for Turbulent Flows 814
11.20.1. Custom Field Functions for Turbulence 822
11.20.2. Postprocessing Turbulent Flow Statistics 822
11.20.3. Troubleshooting 824
12. Modeling Heat Transfer 825
12.1. Introduction 825
12.2. Modeling Conductive and Convective Heat Transfer 825
12.2.1. Solving Heat Transfer Problems 825
12.2.1.1. Limiting the Predicted Temperature Range 827
12.2.1.2. Modeling Heat Transfer in Two Separated Fluid Regions 827
12.2.2. Solution Strategies for Heat Transfer Modeling 828
12.2.2.1. Under-Relaxation of the Energy Equation 828
12.2.2.2. Under-Relaxation of Temperature When the Enthalpy Equation is Solved 828
12.2.2.3. Disabling the Species Diffusion Term 829
12.2.2.4. Step-by-Step Solutions 829
12.2.2.4.1. Decoupled Flow and Heat Transfer Calculations 829
12.2.2.4.2. Coupled Flow and Heat Transfer Calculations 829
12.2.2.5. Specifying a Solid Time step 829
12.2.2.5.1. Automatic Time Step Calculation 830
12.2.3. Postprocessing Heat Transfer Quantities 831

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12.2.3.1. Available Variables for Postprocessing 831
12.2.3.2. Definition of Enthalpy and Energy in Reports and Displays 832
12.2.3.3. Reporting Heat Transfer Through Boundaries 832
12.2.3.4. Reporting Heat Transfer Through a Surface 832
12.2.3.5. Reporting Averaged Heat Transfer Coefficients 832
12.2.3.6. Exporting Heat Flux Data 832
12.2.4. Natural Convection and Buoyancy-Driven Flows 833
12.2.4.1. Modeling Natural Convection in a Closed Domain 833
12.2.4.2. The Boussinesq Model 833
12.2.4.3. Limitations of the Boussinesq Model 834
12.2.4.4. Steps in Solving Buoyancy-Driven Flow Problems 834
12.2.4.5. Operating Density 836
12.2.4.5.1. Setting the Operating Density 836
12.2.4.6. Solution Strategies for Buoyancy-Driven Flows 837
12.2.4.6.1. Guidelines for Solving High-Rayleigh-Number Flows 837
12.2.4.7. Postprocessing Buoyancy-Driven Flows 838
12.2.5. Shell Conduction Considerations 838
12.2.5.1. Introduction 838
12.2.5.2. Physical Treatment 838
12.2.5.3. Limitations of Shell Conduction Walls 839
12.2.5.4. Managing Shell Conduction Walls 840
12.2.5.5. Initializing Shells 843
12.2.5.6. Locking the Temperature for Shells 843
12.2.5.7. Postprocessing Shells 843
12.3. Modeling Radiation 845
12.3.1. Using the Radiation Models 845
12.3.2. Setting Up the P-1 Model with Non-Gray Radiation 847
12.3.3. Setting Up the DTRM 848
12.3.3.1. Defining the Rays 848
12.3.3.2. Controlling the Clusters 849
12.3.3.3. Controlling the Rays 850
12.3.3.4. Writing and Reading the DTRM Ray File 850
12.3.3.5. Displaying the Clusters 851
12.3.4. Setting Up the S2S Model 851
12.3.4.1. View Factors and Clustering Settings 853
12.3.4.1.1. Forming Surface Clusters 853
12.3.4.1.1.1. Setting the Split Angle for Clusters 856
12.3.4.1.2. Setting Up the View Factor Calculation 856
12.3.4.1.2.1. Selecting the Basis for Computing View Factors 856
12.3.4.1.2.2. Selecting the Method for Computing View Factors 857
12.3.4.1.2.3. Accounting for Blocking Surfaces 858
12.3.4.1.2.4. Specifying Boundary Zone Participation 858
12.3.4.2. Computing View Factors 860
12.3.4.2.1. Computing View Factors Inside ANSYS Fluent 860
12.3.4.2.2. Computing View Factors Outside ANSYS Fluent 862
12.3.4.3. Reading View Factors into ANSYS Fluent 863
12.3.5. Setting Up the DO Model 864
12.3.5.1. Angular Discretization 864
12.3.5.2. Defining Non-Gray Radiation for the DO Model 865
12.3.5.3. Enabling DO/Energy Coupling 866
12.3.6. Setting Up the MC Model 867
12.3.7. Defining Material Properties for Radiation 868

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12.3.7.1. Absorption Coefficient for a Non-Gray Model 869
12.3.7.2. Refractive Index for a Non-Gray Model 869
12.3.8. Defining Boundary Conditions for Radiation 869
12.3.8.1. Inlet and Exit Boundary Conditions 869
12.3.8.1.1. Emissivity 869
12.3.8.1.2. Black Body Temperature 869
12.3.8.2. Wall Boundary Conditions for the DTRM, and the P-1, S2S, and Rosseland Models 870
12.3.8.2.1. Boundary Conditions for the S2S Model 870
12.3.8.3. Wall Boundary Conditions for the DO Model 871
12.3.8.3.1. Opaque Walls 871
12.3.8.3.2. Semi-Transparent Walls 873
12.3.8.4. Boundary Conditions for the MC Model 876
12.3.8.5. Solid Cell Zones Conditions for the DO Model 877
12.3.8.6. Thermal Boundary Conditions 877
12.3.9. Solution Strategies for Radiation Modeling 878
12.3.9.1. P-1 Model Solution Parameters 878
12.3.9.2. DTRM Solution Parameters 879
12.3.9.3. S2S Solution Parameters 880
12.3.9.4. DO Solution Parameters 880
12.3.9.5. MC Solution Parameters 880
12.3.9.6. Running the Calculation 880
12.3.9.6.1. Residual Reporting for the P-1 Model 881
12.3.9.6.2. Residual Reporting for the DO Model 881
12.3.9.6.3. Residual Reporting for the DTRM 881
12.3.9.6.4. Residual Reporting for the S2S Model 881
12.3.9.6.5. Disabling the Update of the Radiation Fluxes 881
12.3.10. Postprocessing Radiation Quantities 882
12.3.10.1. Available Variables for Postprocessing 882
12.3.10.2. Reporting Radiative Heat Transfer Through Boundaries 883
12.3.10.3. Overall Heat Balances When Using the DTRM 883
12.3.10.4. Displaying Rays and Clusters for the DTRM 883
12.3.10.4.1. Displaying Clusters 884
12.3.10.4.2. Displaying Rays 884
12.3.10.4.3. Including the Mesh in the Display 885
12.3.10.5. Reporting Radiation in the S2S Model 885
12.3.11. Solar Load Model 886
12.3.11.1. Introduction 887
12.3.11.2. Solar Ray Tracing 887
12.3.11.2.1. Shading Algorithm 888
12.3.11.2.2. Glazing Materials 888
12.3.11.2.3. Inputs 889
12.3.11.3. DO Irradiation 890
12.3.11.4. Solar Calculator 891
12.3.11.4.1. Inputs/Outputs 891
12.3.11.4.2. Theory 892
12.3.11.4.3. Computation of Load Distribution 893
12.3.11.5. Using the Solar Load Model 894
12.3.11.5.1. User-Defined Functions (UDFs) for Solar Load 894
12.3.11.5.2. Setting Up the Solar Load Model 894
12.3.11.5.3. Setting Boundary Conditions for Solar Loading 899
12.3.11.5.4. Solar Ray Tracing 900
12.3.11.5.5. DO Irradiation 905

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12.3.11.5.6. Text Interface-Only Commands 907
12.3.11.5.6.1. Automatically Saving Solar Ray Tracing Data 907
12.3.11.5.6.2. Automatically Reading Solar Data 907
12.3.11.5.6.3. Aligning the Camera Direction With the Position of the Sun 908
12.3.11.5.6.4. Specifying the Scattering Fraction 908
12.3.11.5.6.5. Applying the Solar Load on Adjacent Fluid Cells 908
12.3.11.5.6.6. Specifying Quad Tree Refinement Factor 908
12.3.11.5.6.7. Specifying Ground Reflectivity 909
12.3.11.5.6.8. Additional Text Interface Commands 909
12.3.11.6. Postprocessing Solar Load Quantities 909
12.3.11.6.1. Solar Load Animation at Different Sun Positions 910
12.3.11.6.2. Reporting and Displaying Solar Load Quantities 911
12.4. Modeling Periodic Heat Transfer 912
12.4.1. Overview and Limitations 912
12.4.1.1. Overview 912
12.4.1.2. Constraints for Periodic Heat Transfer Predictions 912
12.4.2. Theory 913
12.4.2.1. Definition of the Periodic Temperature for Constant- Temperature Wall Conditions 913
12.4.2.2. Definition of the Periodic Temperature Change σ for Specified Heat Flux Conditions 913
12.4.3. Using Periodic Heat Transfer 914
12.4.4. Solution Strategies for Periodic Heat Transfer 915
12.4.5. Monitoring Convergence 916
12.4.6. Postprocessing for Periodic Heat Transfer 916
13. Modeling Heat Exchangers 919
13.1. Choosing a Heat Exchanger Model 920
13.2. The Dual Cell Model 921
13.2.1. Restrictions 922
13.2.2. Using the Dual Cell Heat Exchanger Model 922
13.3. The Macro Heat Exchanger Models 930
13.3.1. Restrictions 931
13.3.2. Using the Ungrouped Macro Heat Exchanger Model 932
13.3.2.1. Selecting the Zone for the Heat Exchanger 937
13.3.2.2. Specifying Heat Exchanger Performance Data 937
13.3.2.3. Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions 938
13.3.2.4. Defining the Macros 938
13.3.2.4.1. Viewing the Macros 939
13.3.2.5. Specifying the Auxiliary Fluid Properties and Conditions 940
13.3.2.6. Setting the Pressure-Drop Parameters and Effectiveness 941
13.3.2.6.1. Using the Default Core Porosity Model 941
13.3.2.6.2. Defining a New Core Porosity Model 941
13.3.2.6.3. Reading Heat Exchanger Parameters from an External File 942
13.3.2.6.4. Viewing the Parameters for an Existing Core Model 943
13.3.3. Using the Grouped Macro Heat Exchanger Model 943
13.3.3.1. Selecting the Fluid Zones for the Heat Exchanger Group 949
13.3.3.2. Selecting the Upstream Heat Exchanger Group 949
13.3.3.3. Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions 950
13.3.3.4. Specifying the Auxiliary Fluid Properties 950
13.3.3.5. Specifying Supplementary Auxiliary Fluid Streams 950
13.3.3.6. Initializing the Auxiliary Fluid Temperature 950
13.4. Postprocessing for the Heat Exchanger Model 950
13.4.1. Heat Exchanger Reporting 951
13.4.1.1. Computed Heat Rejection 951

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.4.1.2. **Inlet/Outlet Temperature** 952
13.4.1.3. **Mass Flow Rate** 953
13.4.1.4. **Specific Heat** 954
13.4.2. Total Heat Rejection Rate 954
13.5. Useful Reporting TUI Commands 955
14. Modeling Species Transport and Finite-Rate Chemistry 957
14.1. Volumetric Reactions 958
14.1.1. Overview of User Inputs for Modeling Species Transport and Reactions 958
14.1.1.1. Mixture Materials 959
14.1.2. Enabling Species Transport and Reactions and Choosing the Mixture Material 960
14.1.3. Importing a Volumetric Kinetic Mechanism in CHEMKIN Format 967
14.1.3.1. Using ANSYS Encrypted Mechanisms 968
14.1.3.2. Procedure for Importing Volumetric CHEMKIN Mechanisms 968
14.1.3.3. CHEMKIN Mechanisms Included with ANSYS Fluent 971
14.1.4. Defining Properties for the Mixture and Its Constituent Species 972
14.1.4.1. Defining the Species in the Mixture 973
14.1.4.1.1. Overview of the Species Dialog Box 974
14.1.4.1.2. Adding Species to the Mixture 975
14.1.4.1.3. Removing Species from the Mixture 976
14.1.4.1.4. Reordering Species 976
14.1.4.1.5. The Naming and Ordering of Species 976
14.1.4.2. Defining Reactions 977
14.1.4.2.1. Inputs for Reaction Definition 977
14.1.4.2.2. Defining Species and Reactions for Fuel Mixtures 983
14.1.4.3. Defining Zone-Based Reaction Mechanisms 984
14.1.4.3.1. Inputs for Reaction Mechanism Definition 984
14.1.4.4. Defining Physical Properties for the Mixture 986
14.1.4.5. Defining Physical Properties for the Species in the Mixture 987
14.1.5. Setting up Coal Simulations with the Coal Calculator Dialog Box 988
14.1.6. Defining Cell Zone and Boundary Conditions for Species 990
14.1.6.1. Diffusion at Inlets with the Pressure-Based Solver 990
14.1.7. Defining Other Sources of Chemical Species 990
14.1.8. Solution Procedures for Chemical Mixing and Finite-Rate Chemistry 990
14.1.8.1. Stability and Convergence in Reacting Flows 991
14.1.8.2. Two-Step Solution Procedure (Steady-state Only) 991
14.1.8.3. Density Under-Relaxation 992
14.1.8.4. Ignition in Steady-State Combustion Simulations 992
14.1.8.5. Solution of Stiff Chemistry Systems 992
14.1.8.6. Eddy-Dissipation Concept Model Solution Procedure 993
14.1.9. Postprocessing for Species Calculations 994
14.1.9.1. Averaged Species Concentrations 995
14.2. Wall Surface Reactions and Chemical Vapor Deposition 995
14.2.1. Overview of Surface Species and Wall Surface Reactions 995
14.2.2. Importing a Surface Kinetic Mechanism in CHEMKIN Format 995
14.2.2.1. Compatibility and Limitations for Gas Phase Reactions 998
14.2.2.2. Compatibility and Limitations for Surface Reactions 998
14.2.3. Manual Inputs for Wall Surface Reactions 998
14.2.4. Including Mass Transfer To Surfaces in Continuity 1000
14.2.5. Wall Surface Mass Transfer Effects in the Energy Equation 1000
14.2.6. Modeling the Heat Release Due to Wall Surface Reactions 1000
14.2.7. Solution Procedures for Wall Surface Reactions 1001
14.2.8. Postprocessing for Surface Reactions 1001

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.3. Particle Surface Reactions 1001
14.3.1. User Inputs for Particle Surface Reactions 1001
14.3.2. Modeling Gaseous Solid Catalyzed Reactions 1002
14.3.3. Using the Multiple Surface Reactions Model for Discrete-Phase Particle Combustion 1002
14.4. Electrochemical Reactions 1003
14.4.1. Overview of Electrochemical Reactions 1003
14.4.2. User Inputs for Electrochemical Reactions 1004
14.4.3. Electrochemical Reaction Effects in the Energy Equation 1010
14.4.4. Electrochemical Reaction Effects in the Species Transport Equation 1011
14.4.5. Including Mass Transfer in Continuity 1011
14.4.6. Solution Procedures for Electrochemical Reactions 1011
14.5. Species Transport Without Reactions 1011
14.6. Reacting Channel Model 1012
14.6.1. Overview and Limitations of the Reacting Channel Model 1013
14.6.2. Enabling the Reacting Channel Model 1013
14.6.3. Boundary Conditions for Channel Walls 1017
14.6.4. Postprocessing for Reacting Channel Model Calculations 1018
14.7. Reactor Network Model 1020
14.7.1. Overview and Limitations of the Reactor Network Model 1021
14.7.2. Solving Reactor Networks 1021
14.7.3. Postprocessing Reactor Network Calculations 1024
15. Modeling Non-Premixed Combustion 1025
15.1. Steps in Using the Non-Premixed Model 1025
15.1.1. Preliminaries 1025
15.1.2. Defining the Problem Type 1026
15.1.3. Overview of the Problem Setup Procedure 1026
15.2. Setting Up the Equilibrium Chemistry Model 1031
15.2.1. Choosing Adiabatic or Non-Adiabatic Options 1032
15.2.2. Specifying the Operating Pressure for the System 1033
15.2.3. Enabling a Secondary Inlet Stream 1033
15.2.4. Choosing to Define the Fuel Stream(s) Empirically 1034
15.2.5. Enabling the Rich Flammability Limit (RFL) Option 1035
15.3. Setting Up the Steady and Unsteady Diffusion Flamelet Models 1036
15.3.1. Choosing Adiabatic or Non-Adiabatic Options 1036
15.3.2. Specifying the Operating Pressure for the System 1037
15.3.3. Specifying a Chemical Mechanism File for Flamelet Generation 1037
15.3.4. Importing a Flamelet 1037
15.3.5. Using the Unsteady Diffusion Flamelet Model 1037
15.3.6. Using the Diesel Unsteady Laminar Flamelet Model 1038
15.3.6.1. Recommended Settings for Internal Combustion Engines 1041
15.3.7. Resetting Diesel Unsteady Flamelets 1041
15.4. Defining the Stream Compositions 1042
15.4.1. Setting Boundary Stream Species 1044
15.4.1.1. Including Condensed Species 1045
15.4.2. Modifying the Database 1045
15.4.3. Composition Inputs for Empirically-Defined Fuel Streams 1045
15.4.4. Modeling Liquid Fuel Combustion Using the Non-Premixed Model 1045
15.4.5. Modeling Coal Combustion Using the Non-Premixed Model 1046
15.4.5.1. Defining the Coal Composition: Single-Mixture-Fraction Models 1047
15.4.5.2. Defining the Coal Composition: Two-Mixture-Fraction Models 1048
15.4.5.3. Additional Coal Modeling Inputs in ANSYS Fluent 1049
15.4.5.4. Postprocessing Non-Premixed Models of Coal Combustion 1051

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.4.5.5. The Coal Calculator 1051
15.5. Setting Up Control Parameters 1053
15.5.1. Forcing the Exclusion and Inclusion of Equilibrium Species 1053
15.5.2. Defining the Flamelet Controls 1054
15.5.3. Zeroing Species in the Initial Unsteady Flamelet 1055
15.6. Calculating the Flamelets 1056
15.6.1. Steady Diffusion Flamelet 1056
15.6.2. Unsteady Diffusion Flamelet 1058
15.6.3. Saving the Flamelet Data 1060
15.6.4. Postprocessing the Flamelet Data 1060
15.7. Calculating the Look-Up Tables 1063
15.7.1. Full Tabulation of the Two-Mixture-Fraction Model 1067
15.7.2. Stability Issues in Calculating Chemical Equilibrium Look-Up Tables 1068
15.7.3. Saving the Look-Up Tables 1068
15.7.4. Postprocessing the Look-Up Table Data 1068
15.8. Standard Files for Diffusion Flamelet Modeling 1073
15.8.1. Sample Standard Diffusion Flamelet File 1074
15.8.2. Missing Species 1075
15.9. Setting Up the Inert Model 1075
15.9.1. Setting Boundary Conditions for Inert Transport 1076
15.9.2. Initializing the Inert Stream 1077
15.9.2.1. Inert Fraction 1077
15.9.2.2. Inert Composition 1077
15.9.3. Resetting Inert EGR 1077
15.10. Defining Non-Premixed Boundary Conditions 1079
15.10.1. Input of Mixture Fraction Boundary Conditions 1079
15.10.2. Diffusion at Inlets 1080
15.10.3. Input of Thermal Boundary Conditions and Fuel Inlet Velocities 1080
15.11. Defining Non-Premixed Physical Properties 1080
15.12. Solution Strategies for Non-Premixed Modeling 1081
15.12.1. Single-Mixture-Fraction Approach 1081
15.12.2. Two-Mixture-Fraction Approach 1081
15.12.3. Starting a Non-Premixed Calculation From a Previous Case File 1082
15.12.3.1. Retrieving the PDF File During Case File Reads 1083
15.12.4. Solving the Flow Problem 1083
15.12.4.1. Under-Relaxation Factors for PDF Equations 1083
15.12.4.2. Density Under-Relaxation 1083
15.12.4.3. Tuning the PDF Parameters for Two-Mixture-Fraction Calculations 1084
15.13. Postprocessing the Non-Premixed Model Results 1085
15.13.1. Postprocessing for Inert Calculations 1087
16. Modeling Premixed Combustion 1089
16.1. Overview and Limitations 1089
16.1.1. Limitations of the Premixed Combustion Model 1090
16.2. Using the Premixed Combustion Model 1090
16.2.1. Enabling the Premixed Combustion Model 1091
16.2.2. Choosing an Adiabatic or Non-Adiabatic Model 1092
16.3. Setting Up the C-Equation and G-Equation Models 1092
16.3.1. Modifying the Constants for the Zimont Flame Speed Model 1092
16.3.2. Modifying the Constants for the Peters Flame Speed Model 1093
16.3.3. Additional Options for the G-Equation Model 1093
16.3.4. Defining Physical Properties for the Unburnt Mixture 1093
16.3.5. Setting Boundary Conditions for the Progress Variable 1094

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.3.6. Initializing the Progress Variable 1094
16.4. Setting Up the Extended Coherent Flame Model 1094
16.4.1. Modifying the ECFM Model Variant 1095
16.4.2. Modifying the Constants for the ECFM Flame Speed Closure 1095
16.4.3. Setting Boundary Conditions for the ECFM Transport 1096
16.4.4. Initializing the Flame Area Density 1096
16.5. Postprocessing for Premixed Combustion Calculations 1096
16.5.1. Computing Species Concentrations 1098
17. Modeling Partially Premixed Combustion 1099
17.1. Overview and Limitations 1099
17.1.1. Overview 1099
17.1.2. Limitations 1099
17.2. Using the Partially Premixed Combustion Model 1100
17.2.1. Setup and Solution Procedure 1100
17.2.2. Importing a Flamelet 1102
17.2.3. Flamelet Generated Manifold 1103
17.2.4. Calculating the Look-Up Tables 1105
17.2.4.1. Postprocessing the Look-Up Tables with Flamelet Generated Manifolds 1107
17.2.5. Standard Files for Flamelet Generated Manifold Modeling 1109
17.2.5.1. Sample Standard FGM File 1109
17.2.6. Modifying the Unburnt Mixture Property Polynomials 1110
17.2.7. Modeling In Cylinder Combustion 1112
18. Modeling a Composition PDF Transport Problem 1113
18.1. Overview and Limitations 1113
18.2. Steps for Using the Composition PDF Transport Model 1113
18.3. Enabling the Lagrangian Composition PDF Transport Model 1115
18.4. Enabling the Eulerian Composition PDF Transport Model 1118
18.4.1. Defining Species Boundary Conditions 1119
18.4.1.1. Equilibrating Inlet Streams 1120
18.5. Initializing the Solution 1120
18.6. Monitoring the Solution 1121
18.6.1. Running Unsteady Composition PDF Transport Simulations 1122
18.6.2. Running Compressible Lagrangian PDF Transport Simulations 1123
18.6.3. Running Lagrangian PDF Transport Simulations with Conjugate Heat Transfer 1123
18.7. Postprocessing for Lagrangian PDF Transport Calculations 1123
18.7.1. Reporting Options 1123
18.7.2. Particle Tracking Options 1124
18.8. Postprocessing for Eulerian PDF Transport Calculations 1125
18.8.1. Reporting Options 1125
19. Using Chemistry Acceleration 1127
19.1. Using ISAT 1128
19.1.1. ISAT Parameters 1128
19.1.2. Monitoring ISAT 1129
19.1.3. Using ISAT Efficiently 1130
19.1.4. Reading and Writing ISAT Tables 1130
19.2. Using Dynamic Mechanism Reduction 1131
19.2.1. Mechanism Reduction Parameters 1132
19.2.2. Monitoring and Postprocessing Dynamic Mechanism Reduction 1134
19.2.3. Using Dynamic Mechanism Reduction Effectively 1135
19.3. Using Chemistry Agglomeration 1136
19.4. Dimension Reduction 1136
19.5. Using Dynamic Cell Clustering 1137

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20. Modeling Engine Ignition 1139
20.1. Spark Model 1139
20.1.1. Using the Spark Model 1139
20.1.2. Using the ECFM Spark Model 1141
20.2. Autoignition Models 1142
20.2.1. Using the Autoignition Models 1143
20.3. Crevice Model 1145
20.3.1. Using the Crevice Model 1146
20.3.2. Crevice Model Solution Details 1148
20.3.3. Postprocessing for the Crevice Model 1148
20.3.3.1. Using the Crevice Output File 1150
21. Modeling Pollutant Formation 1153
21.1. NOx Formation 1153
21.1.1. Using the NOx Model 1153
21.1.1.1. Decoupled Analysis: Overview 1153
21.1.1.2. Enabling the NOx Models 1154
21.1.1.3. Defining the Fuel Streams 1156
21.1.1.4. Specifying a User-Defined Function for the NOx Rate 1158
21.1.1.5. Setting Thermal NOx Parameters 1159
21.1.1.6. Setting Prompt NOx Parameters 1159
21.1.1.7. Setting Fuel NOx Parameters 1160
21.1.1.7.1. Setting Gaseous and Liquid Fuel NOx Parameters 1160
21.1.1.7.2. Setting Solid (Coal) Fuel NOx Parameters 1161
21.1.1.8. Setting N2O Pathway Parameters 1162
21.1.1.9. Setting Parameters for NOx Reburn 1163
21.1.1.10. Setting SNCR Parameters 1164
21.1.1.11. Setting Turbulence Parameters 1166
21.1.1.12. Defining Boundary Conditions for the NOx Model 1169
21.1.2. Solution Strategies 1169
21.1.3. Postprocessing 1170
21.2. SOx Formation 1171
21.2.1. Using the SOx Model 1171
21.2.1.1. Enabling the SOx Model 1172
21.2.1.2. Defining the Fuel Streams 1173
21.2.1.3. Defining the SOx Fuel Stream Settings 1175
21.2.1.3.1. Setting SOx Parameters for Gaseous and Liquid Fuel Types 1175
21.2.1.3.2. Setting SOx Parameters for a Solid Fuel 1176
21.2.1.4. Defining the SOx Formation Model Parameters 1178
21.2.1.5. Setting Turbulence Parameters 1179
21.2.1.6. Specifying a User-Defined Function for the SOx Rate 1181
21.2.1.7. Defining Boundary Conditions for the SOx Model 1181
21.2.2. Solution Strategies 1182
21.2.3. Postprocessing 1183
21.3. Soot Formation 1184
21.3.1. Using the Soot Models 1184
21.3.1.1. Setting Up the One-Step Model 1185
21.3.1.2. Setting Up the Two-Step Model 1187
21.3.1.3. Setting Up the Moss-Brookes Model and the Hall Extension 1190
21.3.1.3.1. Specifying a User-Defined Function for the Soot Oxidation Rate 1193
21.3.1.3.2. Specifying a User-Defined Function for the Soot Precursor Concentration 1193
21.3.1.3.3. Species Definition for the Moss-Brookes Model with a User-Defined Precursor Correlation 1194

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

21.3.1.4. Setting Up the Method of Moments Soot Model 1196
21.3.1.5. Defining Boundary Conditions for the Soot Model 1201
21.3.1.6. Reporting Soot Quantities 1201
21.4. Using the Decoupled Detailed Chemistry Model 1202
22. Predicting Aerodynamically Generated Noise 1205
22.1. Overview 1205
22.1.1. Direct Method 1205
22.1.2. Integral Method Based on Acoustic Analogy 1206
22.1.3. Broadband Noise Source Models 1206
22.2. Using the Ffowcs-Williams and Hawkings Acoustics Model 1207
22.2.1. Enabling the FW-H Acoustics Model 1208
22.2.1.1. Setting Model Constants 1209
22.2.1.2. Computing Sound "on the Fly" 1210
22.2.1.3. Writing Source Data Files 1211
22.2.1.3.1. Exporting Source Data Without Enabling the FW-H Model: Using the ANSYS Fluent ASD Format 1212
22.2.1.3.2. Exporting Source Data Without Enabling the FW-H Model: Using the CGNS Format 1213
22.2.2. Specifying Source Surfaces 1214
22.2.2.1. Saving Source Data 1216
22.2.3. Specifying Acoustic Receivers 1217
22.2.4. Specifying the Time Step 1218
22.2.5. Postprocessing the FW-H Acoustics Model Data 1219
22.2.5.1. Writing Acoustic Signals 1219
22.2.5.2. Reading Unsteady Acoustic Source Data 1220
22.2.5.2.1. Pruning the Signal Data Automatically 1221
22.2.5.3. Reporting the Static Pressure Time Derivative 1222
22.2.5.4. Using the FFT Capabilities for Sound Pressure Signals 1222
22.2.6. FFT of Acoustic Sources: Band Analysis and Export of Surface Pressure Spectra 1222
22.2.6.1. Using the FFT of Acoustic Sources 1223
22.3. Using the Broadband Noise Source Models 1231
22.3.1. Enabling the Broadband Noise Source Models 1231
22.3.1.1. Setting Model Constants 1232
22.3.2. Postprocessing the Broadband Noise Source Model Data 1232
23. Modeling Discrete Phase 1235
23.1. Introduction 1235
23.1.1. Concepts 1236
23.1.1.1. Uncoupled vs. Coupled DPM 1236
23.1.1.2. Steady vs. Unsteady Tracking 1236
23.1.1.3. Parcels 1237
23.1.2. Limitations 1238
23.1.2.1. Limitation on the Particle Volume Fraction 1238
23.1.2.2. Limitation on Modeling Continuous Suspensions of Particles 1238
23.1.2.3. Limitations on Modeling Particle Rotation 1238
23.1.2.4. Limitations on Using the Discrete Phase Model with Other ANSYS Fluent Models 1238
23.1.2.5. Limitations on Using the Hybrid Parallel Method 1239
23.1.2.6. Limitations on Using the Lagrangian Wall Film Model 1240
23.2. Steps for Using the Discrete Phase Models 1240
23.2.1. Options for Interaction with the Continuous Phase 1241
23.2.2. Steady/Transient Treatment of Particles 1242
23.2.3. Tracking Parameters for the Discrete Phase Model 1246
23.2.4. Drag Laws 1249

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23.2.5. Physical Models for the Discrete Phase Model 1249
23.2.5.1. Including Radiation Heat Transfer Effects on the Particles 1250
23.2.5.2. Including Thermophoretic Force Effects on the Particles 1251
23.2.5.3. Including Saffman Lift Force Effects on the Particles 1251
23.2.5.4. Including the Virtual Mass Force and Pressure Gradient Effects on Particles 1251
23.2.5.5. Monitoring Erosion/Accretion of Particles at Walls 1251
23.2.5.6. Enabling Pressure Dependent Boiling 1251
23.2.5.7. Including the Effect of Droplet Temperature on Latent Heat 1252
23.2.5.8. Including the Effect of Particles on Turbulent Quantities 1252
23.2.5.9. Including Collision and Droplet Coalescence 1252
23.2.5.10. Including the DEM Collision Model 1253
23.2.5.11. Including Droplet Breakup 1253
23.2.5.12. Modeling Collision Using the DEM Model 1253
23.2.5.12.1. Limitations 1258
23.2.5.12.2. Numeric Recommendations 1258
23.2.6. User-Defined Functions 1258
23.2.7. Numerics of the Discrete Phase Model 1260
23.2.7.1. Numerics for Tracking of the Particles 1261
23.2.7.2. Including Coupled Heat-Mass Solution Effects on the Particles 1262
23.2.7.3. Tracking in a Reference Frame 1263
23.2.7.4. Node Based Averaging of Particle Data 1263
23.2.7.5. Linearized Source Terms 1264
23.2.7.6. Staggering of Particles in Space and Time 1264
23.3. Setting Initial Conditions for the Discrete Phase 1265
23.3.1. Injection Types 1266
23.3.2. Particle Types 1268
23.3.3. Point Properties for Single Injections 1269
23.3.4. Point Properties for Group Injections 1270
23.3.5. Point Properties for Cone Injections 1271
23.3.6. Point Properties for Surface Injections 1274
23.3.6.1. Using the Rosin-Rammler Diameter Distribution Method 1275
23.3.7. Point Properties for Plain-Orifice Atomizer Injections 1276
23.3.8. Point Properties for Pressure-Swirl Atomizer Injections 1277
23.3.9. Point Properties for Air-Blast/Air-Assist Atomizer Injections 1278
23.3.10. Point Properties for Flat-Fan Atomizer Injections 1278
23.3.11. Point Properties for Effervescent Atomizer Injections 1280
23.3.12. Point Properties for File Injections 1280
23.3.13. Point Properties for Condensate Injections 1281
23.3.14. Using the Rosin-Rammler Diameter Distribution Method 1281
23.3.14.1. The Stochastic Rosin-Rammler Diameter Distribution Method 1284
23.3.15. Creating and Modifying Injections 1284
23.3.15.1. Creating Injections 1285
23.3.15.2. Modifying Injections 1285
23.3.15.3. Copying Injections 1286
23.3.15.4. Deleting Injections 1286
23.3.15.5. Listing Injections 1286
23.3.15.6. Reading and Writing Injections 1286
23.3.16. Defining Injection Properties 1286
23.3.17. Specifying Injection-Specific Physical Models 1290
23.3.17.1. Drag Laws 1290
23.3.17.2. Particle Rotation 1291
23.3.17.3. Rough Wall Model 1292

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23.3.17.4. Brownian Motion Effects 1292
23.3.17.5. Breakup 1292
23.3.18. Specifying Turbulent Dispersion of Particles 1294
23.3.18.1. Stochastic Tracking 1294
23.3.18.2. Cloud Tracking 1296
23.3.19. Custom Particle Laws 1296
23.3.20. Defining Properties Common to More than One Injection 1297
23.3.20.1. Modifying Properties 1298
23.3.20.2. Modifying Properties Common to a Subset of Selected Injections 1300
23.3.21. Point Properties for Transient Injections 1300
23.4. Setting Boundary Conditions for the Discrete Phase 1301
23.4.1. Discrete Phase Boundary Condition Types 1301
23.4.2. Default Discrete Phase Boundary Conditions 1304
23.4.3. Coefficients of Restitution 1305
23.4.4. Friction Coefficient 1305
23.4.5. Particle-Wall Impingement Heat Transfer 1306
23.4.6. Film Condensation Model 1307
23.4.7. Wall Boundary Layer Model 1309
23.4.8. Setting Particle Erosion and Accretion Parameters 1309
23.5. Setting Material Properties for the Discrete Phase 1313
23.5.1. Summary of Property Inputs 1313
23.5.2. Setting Discrete-Phase Physical Properties 1316
23.5.2.1. The Concept of Discrete-Phase Materials 1316
23.5.2.1.1. Defining Additional Discrete-Phase Materials 1317
23.5.2.2. Description of the Properties 1317
23.6. Solution Strategies for the Discrete Phase 1325
23.6.1. Performing Trajectory Calculations 1325
23.6.1.1. Uncoupled Calculations 1326
23.6.1.2. Coupled Calculations 1326
23.6.1.2.1. Procedures for a Coupled Two-Phase Flow 1327
23.6.1.2.2. Stochastic Tracking in Coupled Calculations 1328
23.6.1.2.3. Under-Relaxation of the Interphase Exchange Terms 1328
23.6.2. Resetting the Interphase Exchange Terms 1329
23.7. Postprocessing for the Discrete Phase 1329
23.7.1. Displaying of Trajectories 1330
23.7.1.1. Specifying Particles for Display 1332
23.7.1.1.1. Controlling the Particle Tracking Style 1332
23.7.1.1.2. Controlling the Vector Style of Particle Tracks 1334
23.7.1.2. Importing Particle Data 1338
23.7.1.3. Options for Particle Trajectory Plots 1339
23.7.1.4. Particle Filtering 1340
23.7.1.5. Graphical Display for Axisymmetric Geometries 1340
23.7.2. Reporting of Trajectory Fates 1340
23.7.2.1. Trajectory Fates 1341
23.7.2.2. Summary Reports 1341
23.7.2.2.1. Elapsed Time 1343
23.7.2.2.2. Mass Transfer Summary 1343
23.7.2.2.3. Energy Transfer Summary 1343
23.7.2.2.4. Heat Rate and Energy Reporting 1344
23.7.2.2.4.1. Change of Heat and Change of Energy Reporting 1345
23.7.2.2.5. Combusting Particles 1346
23.7.2.2.6. Combusting Particles with the Multiple Surface Reaction Model 1347

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23.7.2.2.7. Multicomponent Particles 1347
23.7.3. Step-by-Step Reporting of Trajectories 1347
23.7.4. Reporting of Current Positions for Unsteady Tracking 1349
23.7.5. Reporting of Interphase Exchange Terms (Discrete Phase Sources) 1351
23.7.6. Reporting of Discrete Phase Variables 1351
23.7.7. Reporting of Unsteady DPM Statistics 1353
23.7.8. Sampling of Trajectories 1355
23.7.9. Histogram Reporting of Samples 1356
23.7.10. Summary Reporting of Current Particles 1358
23.7.11. Postprocessing of Erosion/Accretion Rates 1360
23.8. Parallel Processing for the Discrete Phase Model 1360
24. Modeling Multiphase Flows 1365
24.1. Introduction 1365
24.2. Steps for Using a Multiphase Model 1365
24.2.1. Enabling the Multiphase Model 1367
24.2.2. Choosing Volume Fraction Formulation 1371
24.2.2.1. Interface Modeling Type 1371
24.2.2.2. Spatial Discretization Schemes for Volume Fraction 1373
24.2.2.3. Volume Fraction Limits 1374
24.2.2.4. Expert Options 1374
24.2.3. Solving a Homogeneous Multiphase Flow 1375
24.2.4. The Boussinesq Approximation in Multiphase Flow 1375
24.2.5. Modeling Compressible Flows 1375
24.2.6. Defining the Phases 1377
24.2.7. Including Body Forces 1377
24.2.8. Modeling Multiphase Species Transport 1378
24.2.9. Specifying Heterogeneous Reactions 1380
24.2.10. Including Mass Transfer Effects 1383
24.2.10.1. Mass Transfer Mechanisms 1385
24.2.11. Defining Multiphase Cell Zone and Boundary Conditions 1391
24.2.11.1. Steps for Setting Boundary Conditions 1391
24.2.11.2. Steps for Setting Cell Zone Conditions 1397
24.2.11.3. Boundary and Cell Zone Conditions for the Mixture and the Individual Phases 1398
24.2.11.3.1. VOF Model 1398
24.2.11.3.2. Mixture Model 1399
24.2.11.3.3. Eulerian Model 1401
24.2.11.4. Steps for Copying Cell Zone and Boundary Conditions 1406
24.2.12. Setting Initial Volume Fractions 1407
24.2.12.1. Options for Patching Volume Fraction in the VOF Model 1407
24.3. Setting Up the VOF Model 1408
24.3.1. Solving Steady-State VOF Problems 1409
24.3.2. Guidelines for Using the Multiphase Pseudo Transient Solver 1409
24.3.3. Including Coupled Level Set with the VOF Model 1409
24.3.4. Modeling Open Channel Flows 1410
24.3.4.1. Defining Inlet Groups 1411
24.3.4.2. Defining Outlet Groups 1412
24.3.4.3. Setting the Inlet Group 1412
24.3.4.4. Setting the Outlet Group 1412
24.3.4.5. Determining the Free Surface Level 1413
24.3.4.6. Determining the Bottom Level 1414
24.3.4.7. Specifying the Total Height 1414
24.3.4.8. Determining the Velocity Magnitude 1415

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.3.4.9. Determining the Secondary Phase for the Inlet 1415
24.3.4.10. Choosing the Pressure Specification Method 1416
24.3.4.11. Choosing the Density Interpolation Method 1416
24.3.4.12. Open Channel Flow Compatibility with Velocity Inlet 1417
24.3.4.12.1. Velocity Inlet, Open Channel Flow, Steady-State 1417
24.3.4.12.2. Velocity Inlet, Open Channel Flow, Transient 1417
24.3.4.13. Limitations 1417
24.3.4.14. Recommendations for Setting Up an Open Channel Flow Problem 1418
24.3.5. Modeling Open Channel Wave Boundary Conditions 1419
24.3.5.1. Summary Report and Regime Check 1426
24.3.5.2. Transient Profile Support for Wave Inputs 1428
24.3.5.3. Alternative Stokes Wave Theory Variant 1429
24.3.6. Recommendations for Open Channel Initialization 1429
24.3.6.1. Reporting Parameters for Open Channel Wave BC Option 1432
24.3.7. Numerical Beach Treatment for Open Channels 1432
24.3.7.1. Solution Strategies 1435
24.3.8. Defining the Phases for the VOF Model 1436
24.3.8.1. Defining the Primary Phase 1436
24.3.8.2. Defining a Secondary Phase 1437
24.3.8.3. Including Surface Tension and Adhesion Effects 1438
24.3.8.4. Discretizing Using the Phase Localized Compressive Scheme 1443
24.3.9. Setting Time-Dependent Parameters for the Explicit Volume Fraction Formulation 1445
24.3.10. Modeling Solidification/Melting 1447
24.4. Setting Up the Mixture Model 1447
24.4.1. Defining the Phases for the Mixture Model 1447
24.4.1.1. Defining the Primary Phase 1447
24.4.1.2. Defining a Non-Granular Secondary Phase 1447
24.4.1.3. Defining a Granular Secondary Phase 1448
24.4.1.4. Defining the Interfacial Area Concentration 1451
24.4.1.5. Defining Drag Between Phases 1453
24.4.1.6. Defining the Slip Velocity 1454
24.4.2. Including Mixture Drift Force 1454
24.4.3. Including Cavitation Effects 1455
24.5. Setting Up the Eulerian Model 1455
24.5.1. Additional Guidelines for Eulerian Multiphase Simulations 1455
24.5.2. Defining the Phases for the Eulerian Model 1456
24.5.2.1. Defining the Primary Phase 1456
24.5.2.2. Defining a Non-Granular Secondary Phase 1456
24.5.2.3. Defining a Granular Secondary Phase 1457
24.5.2.4. Defining the Interfacial Area Concentration 1461
24.5.2.5. Defining the Interaction Between Phases 1463
24.5.2.5.1. Specifying the Drag Function 1463
24.5.2.5.1.1. Drag Modification 1466
24.5.2.5.2. Specifying the Restitution Coefficients (Granular Flow Only) 1466
24.5.2.5.3. Including the Lift Force 1466
24.5.2.5.4. Including the Wall Lubrication Force 1467
24.5.2.5.5. Including the Turbulent Dispersion Force 1470
24.5.2.5.6. Including Surface Tension and Wall Adhesion Effects 1473
24.5.2.5.7. Including the Virtual Mass Force 1474
24.5.3. Modeling Turbulence 1474
24.5.3.1. Including Turbulence Interaction Source Terms 1476
24.5.3.2. Customizing the k-ε Multiphase Turbulent Viscosity 1478

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.5.4. Including Heat Transfer Effects 1478
24.5.5. Using an Algebraic Interfacial Area Model 1480
24.5.6. Including the Dense Discrete Phase Model 1481
24.5.6.1. Defining a Granular Discrete Phase 1484
24.5.7. Including the Boiling Model 1486
24.5.8. Including the Multi-Fluid VOF Model 1493
24.6. Setting Up the Wet Steam Model 1495
24.6.1. Using User-Defined Thermodynamic Wet Steam Properties 1495
24.6.2. Writing the User-Defined Wet Steam Property Functions (UDWSPF) 1496
24.6.3. Compiling Your UDWSPF and Building a Shared Library File 1498
24.6.4. Loading the UDWSPF Shared Library File 1499
24.6.5. UDWSPF Example 1500
24.7. Solution Strategies for Multiphase Modeling 1504
24.7.1. Coupled Solution for Eulerian Multiphase Flows 1504
24.7.2. Coupled Solution for VOF and Mixture Multiphase Flows 1506
24.7.3. Selecting the Pressure-Velocity Coupling Method 1507
24.7.3.1. Limitations and Recommendations of the Coupled with Volume Fraction Options for the VOF and Mixture Models 1509
24.7.3.2. Solving N-Phase Volume Fraction Equations 1510
24.7.4. Controlling the Volume Fraction Coupled Solution 1510
24.7.5. VOF Model 1512
24.7.5.1. Setting the Reference Pressure Location 1513
24.7.5.2. Pressure Interpolation Scheme 1513
24.7.5.3. Discretization Scheme Selection 1513
24.7.5.4. High-Order Rhie-Chow Face Flux Interpolation 1514
24.7.5.5. Treatment of Unsteady Terms in Rhie-Chow Face Flux Interpolation 1514
24.7.5.6. Using Unstructured Variant of PRESTO Pressure Scheme 1515
24.7.5.7. Pressure-Velocity Coupling and Under-Relaxation for the Time-dependent Formulations 1515
24.7.5.8. Under-Relaxation for the Steady-State Formulation 1515
24.7.6. Mixture Model 1516
24.7.6.1. Setting the Under-Relaxation Factor for the Slip Velocity 1516
24.7.6.2. Calculating an Initial Solution 1516
24.7.7. Eulerian Model 1516
24.7.7.1. Calculating an Initial Solution 1516
24.7.7.2. Temporarily Ignoring Lift and Virtual Mass Forces 1517
24.7.7.3. Using W-Cycle Multigrid 1517
24.7.7.4. Including the Anisotropic Drag Law 1517
24.7.8. Wet Steam Model 1518
24.7.8.1. Boundary Conditions, Initialization, and Patching 1518
24.7.8.2. Solution Limits for the Wet Steam Model 1518
24.7.8.3. Solution Strategies for the Wet Steam Model 1518
24.8. Multiphase Case Check 1519
24.9. Postprocessing for Multiphase Modeling 1520
24.9.1. Model-Specific Variables 1520
24.9.1.1. VOF Model 1520
24.9.1.2. Mixture Model 1520
24.9.1.3. Eulerian Model 1521
24.9.1.4. Multiphase Species Transport 1522
24.9.1.5. Wet Steam Model 1523
24.9.1.6. Dense Discrete Phase Model 1523
24.9.2. Displaying Velocity Vectors 1524

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.9.3. Reporting Fluxes 1524
24.9.4. Reporting Forces on Walls 1524
24.9.5. Reporting Flow Rates 1524
25. Modeling Solidification and Melting 1527
25.1. Setup Procedure 1527
25.2. Procedures for Modeling Continuous Casting 1530
25.3. Modeling Thermal and Solutal Buoyancy 1531
25.4. Solution Procedure 1532
25.5. Postprocessing 1532
26. Modeling Eulerian Wall Films 1535
26.1. Limitations 1535
26.2. Setting Eulerian Wall Film Model Options 1535
26.3. Setting Eulerian Wall Film Solution Controls 1539
26.4. Setting Eulerian Wall Film Boundary, Initial, and Source Term Conditions 1542
26.5. Postprocessing the Eulerian Wall Film 1545
27. Modeling Electric Potential Field 1547
27.1. Overview and Limitations 1547
27.2. Using the Electric Potential Model 1547
27.3. Postprocessing the Electric Potential Field 1549
28. Using the Solver 1551
28.1. Overview of Using the Solver 1551
28.1.1. Choosing the Solver 1553
28.2. Choosing the Spatial Discretization Scheme 1554
28.2.1. First-Order Accuracy vs. Second-Order Accuracy 1555
28.2.1.1. First-to-Higher Order Blending 1555
28.2.2. Other Discretization Schemes 1556
28.2.3. Choosing the Pressure Interpolation Scheme 1556
28.2.4. Choosing the Density Interpolation Scheme 1557
28.2.5. High Order Term Relaxation (HOTR) 1557
28.2.5.1. Limitations 1559
28.2.6. User Inputs 1559
28.3. Pressure-Based Solver Settings 1561
28.3.1. Choosing the Pressure-Velocity Coupling Method 1561
28.3.1.1. SIMPLE vs. SIMPLEC 1562
28.3.1.2. PISO 1562
28.3.1.3. Fractional Step Method 1562
28.3.1.4. Coupled 1563
28.3.1.5. User Inputs 1563
28.3.2. Setting Under-Relaxation Factors 1564
28.3.2.1. User Inputs 1564
28.3.3. Setting Solution Controls for the Non-Iterative Solver 1566
28.3.3.1. User Inputs 1566
28.4. Density-Based Solver Settings 1570
28.4.1. Changing the Courant Number 1571
28.4.1.1. Courant Numbers for the Density-Based Explicit Formulation 1571
28.4.1.2. Courant Numbers for the Density-Based Implicit Formulation 1571
28.4.1.3. User Inputs 1572
28.4.2. Convective Flux Types 1573
28.4.2.1. User Inputs 1573
28.4.3. Convergence Acceleration for Stretched Meshes (CASM) 1573
28.4.4. Specifying the Explicit Relaxation 1575
28.4.5. Turning On FAS Multigrid 1575

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.4.5.1. Setting Coarse Grid Levels 1576
28.4.5.2. Using Residual Smoothing to Increase the Courant Number 1577
28.5. Setting Algebraic Multigrid Parameters 1577
28.5.1. Specifying the Multigrid Cycle Type 1578
28.5.2. Setting the Termination and Residual Reduction Parameters 1578
28.5.3. Setting the AMG Method and the Stabilization Method 1578
28.5.4. Additional Algebraic Multigrid Parameters 1579
28.5.4.1. Fixed Cycle Parameters 1580
28.5.4.2. Coarsening Parameters 1580
28.5.4.3. Smoother Types 1581
28.5.4.4. Flexible Cycle Parameters 1581
28.5.4.5. Setting the Verbosity 1582
28.5.4.6. Returning to the Default Multigrid Parameters 1583
28.5.5. Setting FAS Multigrid Parameters 1583
28.5.5.1. Combating Convergence Trouble 1583
28.5.5.2. "Industrial-Strength" FAS Multigrid 1584
28.6. Setting Solution Limits 1586
28.6.1. Limiting the Values of Solution Variables 1587
28.6.2. Adjusting the Positivity Rate Limit 1588
28.6.3. Resetting Solution Limits 1588
28.7. Setting Multi-Stage Time-Stepping Parameters 1588
28.7.1. Changing the Multi-Stage Scheme 1588
28.7.1.1. Changing the Coefficients and Number of Stages 1589
28.7.1.2. Controlling Updates to Dissipation and Viscous Stresses 1589
28.7.1.3. Resetting the Multi-Stage Parameters 1590
28.8. Selecting Gradient Limiters 1590
28.9. Initializing the Solution 1591
28.9.1. Initializing the Entire Flow Field Using Standard Initialization 1591
28.9.1.1. Saving and Resetting Initial Values 1593
28.9.2. Patching Values in Selected Cells 1593
28.9.2.1. Using Registers 1595
28.9.2.2. Using Field Functions 1595
28.9.2.3. Using Patching Later in the Solution Process 1595
28.10. Full Multigrid (FMG) Initialization 1595
28.10.1. Steps in Using FMG Initialization 1596
28.10.2. Convergence Strategies for FMG Initialization 1597
28.11. Hybrid Initialization 1597
28.11.1. Steps in Using Hybrid Initialization 1597
28.11.2. Solution Strategies for Hybrid Initialization 1599
28.12. Performing Steady-State Calculations 1600
28.12.1. Updating UDF Profiles 1601
28.12.2. Interrupting Iterations 1601
28.12.3. Resetting Data 1601
28.13. Performing Pseudo Transient Calculations 1601
28.13.1. Setting Pseudo Transient Explicit Relaxation Factors 1602
28.13.1.1. User Inputs 1603
28.13.2. Setting Solution Controls for the Pseudo Transient Method 1604
28.13.3. Solving Pseudo-Transient Flow 1605
28.14. Performing Time-Dependent Calculations 1608
28.14.1. User Inputs for Time-Dependent Problems 1609
28.14.1.1. Additional Inputs 1618
28.14.2. Adaptive Time Stepping 1618

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.14.2.1. The Adaptive Time Stepping Algorithm 1619
28.14.2.2. Specifying Parameters for Adaptive Time Stepping 1619
28.14.2.3. Specifying a User-Defined Time Stepping Method 1621
28.14.3. Variable Time Stepping 1621
28.14.3.1. The Variable Time Stepping Algorithm 1621
28.14.3.2. Specifying Parameters for Variable Time Stepping 1621
28.14.4. Postprocessing for Time-Dependent Problems 1622
28.15. Monitoring Solution Convergence 1624
28.15.1. Monitoring Residuals 1624
28.15.1.1. Definition of Residuals for the Pressure-Based Solver 1624
28.15.1.2. Definition of Residuals for the Density-Based Solver 1626
28.15.1.3. Overview of Using the Residual Monitors Dialog Box 1627
28.15.1.4. Printing and Plotting Residuals 1627
28.15.1.5. Storing Residual History Points 1628
28.15.1.6. Controlling Normalization and Scaling 1628
28.15.1.7. Choosing a Convergence Criterion 1629
28.15.1.8. Modifying Convergence Criteria 1631
28.15.1.9. Disabling Monitoring 1631
28.15.1.10. Plot Parameters 1632
28.15.1.11. Postprocessing Residual Values 1632
28.15.2. Monitoring Statistics 1633
28.15.2.1. Plot Parameters 1635
28.15.3. Monitoring Solution Quantities 1635
28.16. Convergence Conditions 1635
28.16.1. Setting Up the Convergence Conditions Dialog Box 1636
28.17. Executing Commands During the Calculation 1638
28.17.1. Defining Macros 1640
28.17.2. Saving Files During the Calculation 1642
28.18. Automatic Initialization of the Solution and Case Modification 1642
28.18.1. Altering the Solution Initialization and Case Modification after Calculating 1647
28.19. Animating the Solution 1647
28.19.1. Creating an Animation Definition 1647
28.19.1.1. Guidelines for Creating an Animation Definition 1649
28.19.2. Playing an Animation Sequence 1650
28.19.2.1. Modifying the View 1651
28.19.2.2. Modifying the Playback Speed 1651
28.19.2.3. Playing Back an Excerpt 1651
28.19.2.4. "Fast-Forwarding" the Animation 1651
28.19.2.5. Continuous Animation 1651
28.19.2.6. Stopping the Animation 1652
28.19.2.7. Advancing the Animation Frame by Frame 1652
28.19.2.8. Deleting an Animation Sequence 1652
28.19.3. Saving an Animation Sequence 1652
28.19.3.1. Solution Animation File 1652
28.19.3.2. Picture File 1653
28.19.3.3. MPEG File 1653
28.19.4. Reading an Animation Sequence 1654
28.20. Checking Your Case Setup 1654
28.20.1. Automatic Implementation 1655
28.20.2. Manual Implementation 1656
28.20.2.1. Checking the Mesh 1657
28.20.2.2. Checking Model Selections 1659

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.20.2.3. Checking Boundary and Cell Zone Conditions 1661
28.20.2.4. Checking Material Properties 1664
28.20.2.5. Checking the Solver Settings 1665
28.21. Convergence and Stability 1667
28.21.1. Judging Convergence 1668
28.21.2. Step-by-Step Solution Processes 1669
28.21.2.1. Selecting a Subset of the Solution Equations 1669
28.21.2.2. Turning Reactions On and Off 1670
28.21.3. Modifying Algebraic Multigrid Parameters 1670
28.21.4. Modifying the Multi-Stage Parameters 1671
28.21.5. Robustness on Meshes of Poor Quality 1671
28.21.6. Warped-Face Gradient Correction 1674
28.22. Solution Steering 1675
28.22.1. Overview of Solution Steering 1675
28.22.2. Solution Steering Strategy 1675
28.22.2.1. Initialization 1676
28.22.3. Using Solution Steering 1676
29. Adapting the Mesh 1681
29.1. Using Adaption 1681
29.1.1. Adaption Example 1682
29.1.2. Adaption Guidelines 1684
29.2. Boundary Adaption 1685
29.2.1. Performing Boundary Adaption 1685
29.2.1.1. Boundary Adaption Based on Number of Cells 1686
29.2.1.2. Boundary Adaption Based on Normal Distance 1686
29.2.1.3. Boundary Adaption Based on Target Boundary Volume 1687
29.3. Gradient Adaption 1688
29.3.1. Performing Gradient Adaption 1688
29.4. Dynamic Gradient Adaption 1690
29.4.1. Dynamic Gradient Adaption Approach 1690
29.4.1.1. Examples of Dynamic Gradient Adaption 1691
29.5. Isovalue Adaption 1691
29.5.1. Performing Isovalue Adaption 1692
29.6. Region Adaption 1692
29.6.1. Performing Region Adaption 1693
29.7. Volume Adaption 1694
29.7.1. Performing Volume Adaption 1694
29.8. Yplus/Ystar Adaption 1695
29.8.1. Performing Yplus or Ystar Adaption 1695
29.9. Anisotropic Adaption 1697
29.9.1. Limitations of Anisotropic Adaption 1697
29.9.2. Performing Anisotropic Adaption 1697
29.9.3. Boundary Layer Redistribution 1698
29.10. Geometry-Based Adaption 1698
29.10.1. Performing Geometry-Based Adaption 1699
29.11. Registers 1700
29.11.1. Manipulating Adaption Registers 1701
29.11.1.1. Changing Register Types 1701
29.11.1.2. Combining Registers 1702
29.11.1.3. Deleting Registers 1703
29.11.2. Modifying Adaption Marks 1703
29.11.3. Displaying Registers 1704

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

29.11.3.1. Adaption Display Options 1704
29.11.4. Adapting to Registers 1705
29.12. Mesh Adaption Controls 1705
29.12.1. Limiting Adaption by Zone 1707
29.12.2. Limiting Adaption by Cell Volume or Volume Weight 1707
29.12.3. Limiting the Total Number of Cells 1707
29.12.4. Controlling the Levels of Refinement During Hanging Node Adaption 1708
30. Creating Surfaces and Cell Registers for Displaying and Reporting Data 1709
30.1. Using Surfaces 1709
30.1.1. Zone Surfaces 1710
30.1.2. Partition Surfaces 1712
30.1.3. Imprint Surfaces 1714
30.1.4. Point Surfaces 1716
30.1.4.1. Using the Point Tool 1717
30.1.4.1.1. Initializing the Point Tool 1717
30.1.4.1.2. Translating the Point Tool 1718
30.1.4.1.3. Resetting the Point Tool 1718
30.1.5. Line and Rake Surfaces 1718
30.1.5.1. Using the Line Tool 1720
30.1.5.1.1. Initializing the Line Tool 1720
30.1.5.1.2. Translating the Line Tool 1721
30.1.5.1.3. Rotating the Line Tool 1721
30.1.5.1.4. Resizing the Line Tool 1722
30.1.5.1.5. Resetting the Line Tool 1722
30.1.6. Plane Surfaces 1722
30.1.6.1. Using the Plane Tool 1724
30.1.6.1.1. Initializing the Plane Tool 1725
30.1.6.1.2. Translating the Plane Tool 1725
30.1.6.1.3. Rotating the Plane Tool 1725
30.1.6.1.4. Resizing the Plane Tool 1726
30.1.6.1.5. Resetting the Plane Tool 1726
30.1.7. Quadric Surfaces 1726
30.1.8. Isosurfaces 1728
30.1.9. Clipping Surfaces 1730
30.1.10. Transforming Surfaces 1732
30.1.11. Grouping, Renaming, and Deleting Surfaces 1734
30.1.11.1. Grouping Surfaces 1735
30.1.11.2. Renaming Surfaces 1736
30.1.11.3. Deleting Surfaces 1736
30.1.11.4. Surface Statistics 1736
30.2. Using Cell Registers 1736
30.2.1. Region 1737
30.2.2. Boundary 1738
30.2.3. Variable Limiter 1739
30.2.4. Field Variable 1739
30.2.5. Residuals 1740
30.2.6. Volume 1741
30.2.7. Yplus/Ystar 1742
30.2.8. Manage Cell Registers 1743
30.2.9. Cell Register Operations 1744
31. Displaying Graphics 1747
31.1. Basic Graphics Generation 1747

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.1.1. Displaying the Mesh 1748
31.1.1.1. Generating Mesh or Outline Plots 1750
31.1.1.2. Mesh and Outline Display Options 1752
31.1.1.2.1. Modifying the Mesh Colors 1752
31.1.1.2.2. Adding Features to an Outline Display 1753
31.1.1.2.3. Drawing Partition Boundaries 1754
31.1.1.2.4. Shrinking Faces and Cells in the Display 1754
31.1.1.3. Creating and Using Mesh Plot Definitions 1755
31.1.2. Displaying Contours and Profiles 1756
31.1.2.1. Generating Contour and Profile Plots 1757
31.1.2.2. Contour and Profile Plot Options 1759
31.1.2.2.1. Drawing Filled Contours or Profiles 1759
31.1.2.2.2. Specifying the Range of Magnitudes Displayed 1759
31.1.2.2.3. Including the Mesh in the Contour Plot 1761
31.1.2.2.4. Choosing Node or Cell Values 1761
31.1.2.2.5. Storing Contour Plot Settings 1762
31.1.2.3. Creating and Using Contour Plot Definitions 1762
31.1.3. Displaying Vectors 1764
31.1.3.1. Generating Vector Plots 1765
31.1.3.2. Displaying Relative Velocity Vectors 1766
31.1.3.3. Vector Plot Options 1766
31.1.3.3.1. Scaling the Vectors 1766
31.1.3.3.2. Skipping Vectors 1767
31.1.3.3.3. Drawing Vectors in the Plane of the Surface 1767
31.1.3.3.4. Displaying Fixed-Length Vectors 1768
31.1.3.3.5. Displaying Vector Components 1768
31.1.3.3.6. Specifying the Range of Magnitudes Displayed 1768
31.1.3.3.7. Changing the Scalar Field Used for Coloring the Vectors 1768
31.1.3.3.8. Displaying Vectors Using a Single Color 1769
31.1.3.3.9. Including the Mesh in the Vector Plot 1769
31.1.3.3.10. Changing the Arrow Characteristics 1769
31.1.3.4. Creating and Managing Custom Vectors 1769
31.1.3.4.1. Creating Custom Vectors 1769
31.1.3.4.2. Manipulating, Saving, and Loading Custom Vectors 1770
31.1.3.5. Creating and Using Vector Plot Definitions 1771
31.1.4. Displaying Pathlines 1772
31.1.4.1. Steps for Generating Pathlines 1772
31.1.4.2. Options for Pathline Plots 1773
31.1.4.2.1. Including the Mesh in the Pathline Display 1774
31.1.4.2.2. Controlling the Pathline Style 1774
31.1.4.2.3. Controlling Pathline Colors 1775
31.1.4.2.4. "Thinning" Pathlines 1775
31.1.4.2.5. Coarsening Pathlines 1775
31.1.4.2.6. Reversing the Pathlines 1776
31.1.4.2.7. Plotting Oil-Flow Pathlines 1776
31.1.4.2.8. Controlling the Pulse Mode 1776
31.1.4.2.9. Controlling the Accuracy 1776
31.1.4.2.10. Plotting Relative Pathlines 1776
31.1.4.2.11. Generating an XY Plot Along Pathline Trajectories 1776
31.1.4.2.12. Saving Pathline Data 1777
31.1.4.2.12.1. Standard Type 1778
31.1.4.2.12.2. Geometry Type 1778

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.1.4.2.12.3. EnSight Type 1779
31.1.4.2.13. Choosing Node or Cell Values 1780
31.1.4.3. Creating and Using Pathline Definitions 1780
31.1.5. Displaying a Scene 1780
31.1.5.1. Generating a Scene 1781
31.1.6. Displaying Results on a Sweep Surface 1782
31.1.6.1. Steps for Generating a Plot Using a Sweep Surface 1782
31.1.6.2. Animating a Sweep Surface Display 1783
31.1.7. Hiding the Graphics Window Display 1783
31.2. Customizing the Graphics Display 1784
31.2.1. Advanced Graphics Overlays 1785
31.2.2. Opening Multiple Graphics Windows 1786
31.2.2.1. Setting the Active Window 1787
31.2.3. Changing the Legend Display 1788
31.2.3.1. Enabling/Disabling the Legend, Logo, and Color Scale 1788
31.2.3.2. Editing the Legend 1788
31.2.3.3. Adding a Title to the Caption 1788
31.2.3.4. Enabling/Disabling the Axes 1789
31.2.3.5. Enabling/Disabling the Ruler 1789
31.2.3.6. Modifying and Displaying/Hiding the Logo 1789
31.2.3.7. Colormap Alignment 1789
31.2.4. Adding Text to the Graphics Window 1789
31.2.4.1. Adding Text Using the Annotate Dialog Box 1790
31.2.4.2. Editing Existing Annotation Text 1790
31.2.4.3. Clearing Annotation Text 1791
31.2.5. Changing the Colormap 1791
31.2.5.1. Predefined Colormaps 1792
31.2.5.2. Selecting a Colormap 1793
31.2.5.2.1. Specifying the Colormap Size and Scale 1793
31.2.5.2.2. Changing the Number Format 1793
31.2.5.3. Displaying Colormap Label 1793
31.2.5.4. Creating a Customized Colormap 1795
31.2.6. Adding Lights 1796
31.2.6.1. Turning on Lighting Effects with the Display Options Dialog Box 1796
31.2.6.2. Turning on Lighting Effects with the **Lights** Dialog Box 1797
31.2.6.3. Defining Light Sources 1797
31.2.6.3.1. Removing a Light 1798
31.2.6.3.2. Resetting the Light Definitions 1798
31.2.7. Modifying the Rendering Options 1798
31.2.7.1. Graphics Device Information 1800
31.3. Controlling the Mouse Button Functions 1800
31.3.1. Button Functions 1800
31.3.2. Modifying the Mouse Button Functions 1801
31.4. Viewing the Application Window 1802
31.5. Modifying the View 1803
31.5.1. Selecting a View 1803
31.5.2. Manipulating the Display 1804
31.5.2.1. Scaling and Centering 1805
31.5.2.2. Rotating the Display 1805
31.5.2.2.1. Spinning the Display with the Mouse 1806
31.5.2.3. Translating the Display 1807
31.5.2.4. Zooming the Display 1807

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.5.3. Controlling Perspective and Camera Parameters 1808
31.5.3.1. Perspective and Orthographic Views 1808
31.5.3.2. Modifying Camera Parameters 1808
31.5.4. Saving and Restoring Views 1809
31.5.4.1. Restoring the Default View 1809
31.5.4.2. Returning to Previous Views 1810
31.5.4.3. Saving Views 1810
31.5.4.4. Reading View Files 1810
31.5.4.5. Deleting Views 1811
31.5.5. Mirroring and Periodic Repeats 1811
31.5.5.1. Periodic Repeats for Graphics 1813
31.5.5.2. Mirroring for Graphics 1815
31.6. Advanced Scene Composition 1815
31.6.1. Selecting the Object(s) to be Manipulated 1817
31.6.2. Changing an Object's Display Properties 1817
31.6.2.1. Controlling Visibility 1818
31.6.2.2. Controlling Object Color and Transparency 1818
31.6.3. Transforming Geometric Objects in a Scene 1819
31.6.3.1. Translating Objects 1820
31.6.3.2. Rotating Objects 1820
31.6.3.3. Scaling Objects 1820
31.6.3.4. Displaying the Meridional View 1820
31.6.4. Modifying Iso-Values 1821
31.6.4.1. Steps for Modifying Iso-Values 1821
31.6.4.2. An Example of Iso-Value Modification for an Animation 1821
31.6.5. Modifying Pathline Attributes 1822
31.6.5.1. An Example of Pathline Modification for an Animation 1822
31.6.6. Deleting an Object from the Scene 1823
31.6.7. Adding a Bounding Frame 1823
31.7. Animating Graphics 1825
31.7.1. Creating an Animation 1826
31.7.1.1. Deleting Key Frames 1826
31.7.2. Playing an Animation 1826
31.7.2.1. Playing Back an Excerpt 1827
31.7.2.2. "Fast-Forwarding" the Animation 1827
31.7.2.3. Continuous Animation 1827
31.7.2.4. Stopping the Animation 1827
31.7.2.5. Advancing the Animation Frame by Frame 1828
31.7.3. Saving an Animation 1828
31.7.3.1. Animation File 1828
31.7.3.2. Picture File 1828
31.7.3.3. MPEG File 1829
31.7.4. Reading an Animation File 1829
31.7.5. Notes on Animation 1829
31.8. Creating Videos 1829
31.8.1. Recording Animations To Video 1830
31.8.1.1. Computer Image vs. Video Image 1830
31.8.1.2. Real-Time vs. Frame-By-Frame 1830
31.8.2. Equipment Required 1831
31.8.3. Recording an Animation with ANSYS Fluent 1832
31.8.3.1. Create an Animation 1832
31.8.3.2. Open a Connection to the VTR Controller 1833

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.8.3.3. Set Up Your Recording Session 1833
31.8.3.3.1. Select the Recording Source 1835
31.8.3.3.2. Choose Real-Time or Frame-By-Frame Recording 1835
31.8.3.3.3. Set the Video Frame Hold Counts 1836
31.8.3.4. Check the Picture Quality 1836
31.8.3.5. Make Sure Your Tape is Formatted (Preblacked) 1837
31.8.3.6. Start the Recording Session 1837
31.9. Histogram and XY Plots 1838
31.9.1. Plot Types 1838
31.9.1.1. XY Plots 1838
31.9.1.2. Histograms 1839
31.9.2. XY Plots of Solution Data 1840
31.9.2.1. Steps for Generating Solution XY Plots 1840
31.9.2.2. Options for Solution XY Plots 1843
31.9.2.2.1. Including External Data in the Solution XY Plot 1844
31.9.2.2.2. Choosing Node or Cell Values 1844
31.9.2.2.3. Saving the Plot Data to a File 1844
31.9.3. XY Plots of File Data 1844
31.9.3.1. Steps for Generating XY Plots of Data in External Files 1844
31.9.3.2. Options for File XY Plots 1845
31.9.3.2.1. Changing the Plot Title 1845
31.9.3.2.2. Changing the Legend Entry 1845
31.9.3.2.3. Changing the Legend Title 1846
31.9.4. XY Plots of Profiles 1846
31.9.4.1. Steps for Generating Plots of Profile Data 1846
31.9.4.2. Steps for Generating Plots of Interpolated Profile Data 1847
31.9.5. XY Plots of Circumferential Averages 1848
31.9.5.1. Steps for Generating an XY Plot of Circumferential Averages 1848
31.9.5.2. Customizing the Appearance of the Plot 1850
31.9.6. XY Plot File Format 1850
31.9.7. Residual Plots 1851
31.9.8. Histograms 1851
31.9.8.1. Steps for Generating Histogram Plots 1851
31.9.8.2. Options for Histogram Plots 1852
31.9.8.2.1. Specifying the Range of Values Plotted 1852
31.9.9. Modifying Axis Attributes 1852
31.9.9.1. Using the Axes Dialog Box 1853
31.9.9.1.1. Changing the Axis Label 1853
31.9.9.1.2. Changing the Format of the Data Labels 1853
31.9.9.1.3. Choosing Logarithmic or Decimal Scaling 1854
31.9.9.1.4. Resetting the Range of the Axis 1854
31.9.9.1.5. Controlling the Major and Minor Rules 1854
31.9.10. Modifying Curve Attributes 1854
31.9.10.1. Using the Curves Dialog Box 1855
31.9.10.1.1. Changing the Line Style 1855
31.9.10.1.2. Changing the Marker Style 1856
31.9.10.1.3. Previewing the Curve Style 1856
31.10. Turbomachinery Postprocessing 1856
31.10.1. Defining the Turbomachinery Topology 1856
31.10.1.1. Boundary Types 1858
31.10.2. Generating Reports of Turbomachinery Data 1859
31.10.2.1. Computing Turbomachinery Quantities 1861

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.10.2.1.1. Mass Flow 1861
31.10.2.1.2. Swirl Number 1861
31.10.2.1.3. Average Total Pressure 1861
31.10.2.1.4. Average Total Temperature 1862
31.10.2.1.5. Average Flow Angles 1862
31.10.2.1.6. Passage Loss Coefficient 1863
31.10.2.1.7. Axial Force 1863
31.10.2.1.8. Torque 1863
31.10.2.1.9. Efficiencies for Pumps and Compressors 1864
31.10.2.1.9.1. Incompressible Flows 1864
31.10.2.1.9.2. Compressible Flows 1865
31.10.2.1.10. Efficiencies for Turbines 1866
31.10.2.1.10.1. Incompressible Flows 1866
31.10.2.1.10.2. Compressible Flows 1866
31.10.3. Displaying Turbomachinery Averaged Contours 1867
31.10.3.1. Steps for Generating Turbomachinery Averaged Contour Plots 1868
31.10.3.2. Contour Plot Options 1869
31.10.4. Displaying Turbomachinery 2D Contours 1869
31.10.4.1. Steps for Generating Turbo 2D Contour Plots 1869
31.10.4.2. Contour Plot Options 1870
31.10.5. Generating Averaged XY Plots of Turbomachinery Solution Data 1871
31.10.5.1. Steps for Generating Turbo Averaged XY Plots 1871
31.10.6. Globally Setting the Turbomachinery Topology 1872
31.10.7. Turbomachinery-Specific Variables 1872
31.11. Fast Fourier Transform (FFT) Postprocessing 1873
31.11.1. Limitations of the FFT Algorithm 1873
31.11.2. Windowing 1873
31.11.3. Fast Fourier Transform (FFT) 1874
31.11.4. Using the FFT Utility 1875
31.11.4.1. Loading Data for Spectral Analysis 1876
31.11.4.2. Customizing the Input and Defining the Spectrum Smoothing 1877
31.11.4.2.1. Customizing the Input Signal Data Set 1878
31.11.4.2.2. Spectrum Smoothing Through Signal Segmentation 1878
31.11.4.2.3. Viewing Data Statistics 1878
31.11.4.2.4. Customizing Titles and Labels 1879
31.11.4.2.5. Applying the Changes in the Input Signal Data 1879
31.11.4.3. Customizing the Output 1879
31.11.4.3.1. Specifying a Function for the Axis 1879
31.11.4.3.2. Specifying a Function for the X Axis 1881
31.11.4.3.3. Specifying Output Options 1882
31.11.4.3.4. Specifying a Windowing Technique 1883
31.11.4.3.5. Specifying Labels and Titles 1883
32. Reporting Alphanumeric Data 1885
32.1. Reporting Conventions 1885
32.2. Monitoring and Reporting Solution Data 1885
32.2.1. Creating Report Definitions 1886
32.2.1.1. Surface Report definitions 1889
32.2.1.2. Volume Report definitions 1891
32.2.1.3. Force and Moment Report definitions 1893
32.2.1.4. Flux Report Definition 1897
32.2.1.5. DPM Report Definition 1899
32.2.1.6. User Defined Report Definition 1901

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.2.1.6.1. User Defined Report Definition Function 1901
32.2.1.6.2. User Defined Report Definition Function Hooking 1902
32.2.1.7. Expression Report Definition 1902
32.2.2. Report Files and Report Plots 1903
32.2.2.1. Creating Report Files 1904
32.2.2.2. Creating Report Plots 1906
32.3. Creating Output Parameters 1909
32.4. Fluxes Through Boundaries 1911
32.4.1. Generating a Flux Report 1911
32.4.2. Flux Reporting for Reacting Flows 1913
32.4.2.1. Flux Reporting with Particles 1914
32.4.2.2. Flux Reporting with Multiphase 1915
32.4.2.3. Flux Reporting with Other Volumetric Sources 1916
32.5. Forces on Boundaries 1916
32.5.1. Generating a Force, Moment, or Center of Pressure Report 1916
32.5.1.1. Example 1918
32.6. Projected Surface Area Calculations 1919
32.7. Surface Integration 1920
32.7.1. Generating a Surface Integral Report 1920
32.8. Volume Integration 1922
32.8.1. Generating a Volume Integral Report 1922
32.9. Histogram Reports 1924
32.10. Discrete Phase 1924
32.11. S2S Information 1924
32.12. Reference Values 1924
32.12.1. Setting Reference Values 1925
32.12.2. Setting the Reference Zone 1927
32.13. Summary Reports of Case Settings 1927
32.13.1. Generating a Summary Report 1927
32.14. System Resource Usage 1927
32.14.1. Processor Information 1928
32.14.2. Memory Information 1929
32.14.3. Process and Model Timers 1929
33. Field Function Definitions 1931
33.1. Node, Cell, and Facet Values 1931
33.1.1. Cell Values 1931
33.1.2. Node Values 1931
33.1.2.1. Vertex Values for Points That are Not Mesh Nodes 1932
33.1.3. Facet Values 1932
33.1.3.1. Facet Values on Zone Surfaces 1932
33.1.3.2. Facet Values on Postprocessing Surfaces 1933
33.2. Velocity Reporting Options 1933
33.3. Field Variables Listed by Category 1934
33.4. Alphabetical Listing of Field Variables and Their Definitions 1957
33.5. Custom Field Functions 2003
33.5.1. Creating a Custom Field Function 2004
33.5.1.1. Using the Calculator Buttons 2005
33.5.1.2. Using the Field Functions List 2005
33.5.2. Manipulating, Saving, and Loading Custom Field Functions 2006
33.5.3. Sample Custom Field Functions 2007
34. Parallel Processing 2009
34.1. Introduction to Parallel Processing 2009

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

34.1.1. Recommended Usage of Parallel ANSYS Fluent 2011
34.2. Starting Parallel ANSYS Fluent Using Fluent Launcher 2012
34.2.1. Setting Parallel Scheduler Options in Fluent Launcher 2015
34.2.2. Setting Additional Options When Running on Remote Linux Machines 2017
34.2.2.1. Setting Job Scheduler Options When Running on Remote Linux Machines 2020
34.3. Starting Parallel ANSYS Fluent on a Windows System 2021
34.3.1. Starting Parallel ANSYS Fluent on a Windows System Using Command Line Options 2021
34.3.1.1. Starting Parallel ANSYS Fluent with the Microsoft Job Scheduler 2023
34.4. Starting Parallel ANSYS Fluent on a Linux System 2026
34.4.1. Starting Parallel ANSYS Fluent on a Linux System Using Command Line Options 2026
34.4.2. Setting Up Your Remote Shell and Secure Shell Clients 2029
34.4.2.1. Configuring the rsh Client 2029
34.4.2.2. Configuring the ssh Client 2029
34.5. Mesh Partitioning and Load Balancing 2030
34.5.1. Overview of Mesh Partitioning 2030
34.5.2. Partitioning the Mesh Automatically 2031
34.5.2.1. Reporting During Auto Partitioning 2033
34.5.3. Partitioning the Mesh Manually and Balancing the Load 2033
34.5.3.1. Guidelines for Partitioning the Mesh 2033
34.5.4. Using the Partitioning and Load Balancing Dialog Box 2033
34.5.4.1. Partitioning 2033
34.5.4.1.1. Example of Setting Selected Registers to Specified Partition IDs 2040
34.5.4.1.2. Partitioning Within Zones or Registers 2043
34.5.4.1.3. Reporting During Partitioning 2043
34.5.4.1.4. Resetting the Partition Parameters 2044
34.5.4.2. Load Balancing 2044
34.5.5. Mesh Partitioning Methods 2046
34.5.5.1. Partition Methods 2046
34.5.5.2. Optimizations 2050
34.5.5.3. Pretesting 2051
34.5.5.4. Using the Partition Filter 2052
34.5.6. Checking the Partitions 2052
34.5.6.1. Interpreting Partition Statistics 2052
34.5.6.2. Examining Partitions Graphically 2055
34.5.7. Load Distribution 2055
34.5.8. Troubleshooting 2056
34.6. Using General Purpose Graphics Processing Units (GPGPUs) With the Algebraic Multigrid (AMG) Solver 2056
34.6.1. Requirements 2057
34.6.2. Limitations 2057
34.6.3. Using and Managing GPGPUs 2058
34.7. Controlling the Threads 2058
34.8. Checking Network Connectivity 2059
34.9. Checking and Improving Parallel Performance 2060
34.9.1. Parallel Check 2060
34.9.2. Checking Parallel Performance 2060
34.9.2.1. Checking Latency and Bandwidth 2062
34.9.3. Optimizing the Parallel Solver 2064
34.9.3.1. Increasing the Report Interval 2064
34.9.3.2. Accelerating View Factor Calculations for General Purpose Computing on Graphics Processing Units (GPGPUs) 2064
34.9.3.3. Accelerating Discrete Ordinates (DO) Radiation Calculations 2065

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

34.9.4. Clearing the Linux File Cache Buffers 2066
35. Design Analysis and Optimization 2067
35.1. The Adjoint Solver 2067
35.1.1. General Observables 2069
35.1.2. General Operations 2072
35.1.3. Discrete Versus Continuous Adjoint Solver 2073
35.1.4. Discrete Adjoint Solver Overview 2074
35.1.5. Adjoint Solver Stabilization 2077
35.1.6. Solution-Based Adaption 2078
35.1.7. Using The Data To Improve A Design 2079
35.1.7.1. Smoothing and Mesh Morphing 2080
35.2. Using the Adjoint Solver 2081
35.2.1. Model Considerations for Using Adjoint Solver 2082
35.2.1.1. Basic Assumptions and Consistency Checks 2082
35.2.1.2. User-Defined Sources 2083
35.2.2. Defining Observables 2084
35.2.2.1. Creating New Observables 2085
35.2.2.2. Editing Observable Definitions 2087
35.2.2.3. Selecting an Observable for Sensitivity Calculation 2090
35.2.3. Solving the Adjoint 2090
35.2.3.1. Using the Adjoint Solution Methods Dialog Box 2091
35.2.3.2. Using the Adjoint Solution Controls Dialog Box 2092
35.2.3.2.1. Stabilized Scheme Settings 2094
35.2.3.2.1.1. Modal Stabilization Scheme 2094
35.2.3.2.1.2. Spatial Stabilization Scheme 2097
35.2.3.2.1.3. Dissipation Scheme 2099
35.2.3.3. Working with Adjoint Residual Monitors 2100
35.2.3.4. Running the Adjoint Calculation 2101
35.2.4. Postprocessing of Adjoint Solutions 2101
35.2.4.1. Field Data 2102
35.2.4.2. Scalar Data 2106
35.2.5. Modifying the Geometry Using the Design Tool 2107
35.2.5.1. Defining the Region for the Design Change 2108
35.2.5.2. Defining Region Conditions 2110
35.2.5.3. Exporting Sensitivity Data 2111
35.2.5.4. Defining Observable Objectives 2112
35.2.5.5. Defining Conditions for the Deformation 2112
35.2.5.6. Shape Modification 2117
35.2.5.7. Design Tool Numerics 2120
35.3. The Mesh Morpher/Optimizer 2122
35.3.1. Limitations 2122
35.3.2. The Optimization Process 2122
35.3.3. Optimizers 2123
35.3.3.1. The Compass Optimizer 2123
35.3.3.2. The NEWUOA Optimizer 2123
35.3.3.3. The Simplex Optimizer 2124
35.3.3.4. The Torczon Optimizer 2124
35.3.3.5. The Powell Optimizer 2124
35.3.3.6. The Rosenbrock Optimizer 2125
35.4. Using the Mesh Morpher/Optimizer 2125
36. Customizing Fluent 2151
36.1. Customizing Fluent with ANSYS ACT 2151

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.1.1. ACT Start Page: Accessing ACT Functionality 2151
36.1.2. Extension Manager: Working with Extensions 2151
36.1.3. Wizards Page: Working with Wizards 2152
36.1.3.1. Fluent Wizard Example: Simple Analysis 2152
36.1.3.1.1. XML Extension Definition File 2152
36.1.3.1.2. IronPython Script 2155
37. Task Page Reference Guide 2159
37.1. Meshing Task Page 2159
37.2. Solution Setup Task Page 2159
37.3. General Task Page 2160
37.3.1. Scale Mesh Dialog Box 2162
37.3.2. Mesh Display Dialog Box 2163
37.3.3. Set Units Dialog Box 2166
37.3.4. Define Unit Dialog Box 2167
37.3.5. Mesh Colors Dialog Box 2168
37.4. Models Task Page 2169
37.4.1. Multiphase Model Dialog Box 2171
37.4.2. Energy Dialog Box 2175
37.4.3. Viscous Model Dialog Box 2176
37.4.4. Radiation Model Dialog Box 2190
37.4.5. View Factors and Clustering Dialog Box 2194
37.4.6. Participating Boundary Zones Dialog Box 2197
37.4.7. Solar Calculator Dialog Box 2199
37.4.8. Heat Exchanger Model Dialog Box 2200
37.4.9. Dual Cell Heat Exchanger Dialog Box 2201
37.4.10. Set Dual Cell Heat Exchanger Dialog Box 2202
37.4.11. Heat Transfer Data Table Dialog Box 2204
37.4.12. NTU Table Dialog Box 2205
37.4.13. Copy From Dialog Box 2206
37.4.14. Ungrouped Macro Heat Exchanger Dialog Box 2207
37.4.15. Velocity Effectiveness Curve Dialog Box 2210
37.4.16. Core Porosity Model Dialog Box 2211
37.4.17. Macro Heat Exchanger Group Dialog Box 2212
37.4.18. Species Model Dialog Box 2216
37.4.19. Coal Calculator Dialog Box 2232
37.4.20. Integration Parameters Dialog Box 2235
37.4.21. Flamelet 3D Surfaces Dialog Box 2237
37.4.22. Flamelet 2D Curves Dialog Box 2239
37.4.23. Unsteady Flamelet Parameters Dialog Box 2239
37.4.24. Flamelet Fluid Zones Dialog Box 2240
37.4.25. PDF Table Dialog Box 2240
37.4.26. Spark Ignition Dialog Box 2243
37.4.27. Set Spark Ignition Dialog Box 2244
37.4.28. Autoignition Model Dialog Box 2246
37.4.29. Inert Dialog Box 2249
37.4.30. NOx Model Dialog Box 2250
37.4.31. SOx Model Dialog Box 2258
37.4.32. Soot Model Dialog Box 2262
37.4.33. Sticking Coefficients Dialog Box 2269
37.4.34. Mechanism Dialog Box 2270
37.4.35. Reactor Network Dialog Box 2271
37.4.36. Decoupled Detailed Chemistry Dialog Box 2273

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.4.37. Reacting Channel Model Dialog Box 2274
37.4.38. Reacting Channel 2D Curves Dialog Box 2276
37.4.39. Discrete Phase Model Dialog Box 2278
37.4.40. DEM Collisions Dialog Box 2285
37.4.41. Create Collision Partner Dialog Box 2286
37.4.42. Copy Collision Partner Dialog Box 2286
37.4.43. Rename Collision Partner Dialog Box 2287
37.4.44. DEM Collision Settings Dialog Box 2287
37.4.45. Solidification and Melting Dialog Box 2288
37.4.46. Acoustics Model Dialog Box 2289
37.4.47. Acoustic Sources Dialog Box 2292
37.4.48. Acoustic Receivers Dialog Box 2293
37.4.49. Interior Cell Zone Selection Dialog Box 2294
37.4.50. Eulerian Wall Film Dialog Box 2294
37.4.51. Potential Dialog Box 2300
37.5. Materials Task Page 2300
37.5.1. Create/Edit Materials Dialog Box 2302
37.5.2. Fluent Database Materials Dialog Box 2312
37.5.3. Open Database Dialog Box 2313
37.5.4. User-Defined Database Materials Dialog Box 2313
37.5.5. Copy Case Material Dialog Box 2315
37.5.6. Material Properties Dialog Box 2315
37.5.7. Edit Property Methods Dialog Box 2316
37.5.8. New Material Name Dialog Box 2317
37.5.9. Polynomial Profile Dialog Box 2318
37.5.10. Piecewise-Linear Profile Dialog Box 2319
37.5.11. Piecewise-Polynomial Profile Dialog Box 2319
37.5.12. Convection/Diffusion Model Dialog Box 2320
37.5.13. Compressible Liquid Dialog Box 2321
37.5.14. User-Defined Functions Dialog Box 2321
37.5.15. Sutherland Law Dialog Box 2322
37.5.16. Power Law Dialog Box 2323
37.5.17. Non-Newtonian Power Law Dialog Box 2324
37.5.18. Carreau Model Dialog Box 2324
37.5.19. Cross Model Dialog Box 2325
37.5.20. Herschel-Bulkley Dialog Box 2326
37.5.21. Biaxial Conductivity Dialog Box 2327
37.5.22. Cylindrical Orthotropic Conductivity Dialog Box 2328
37.5.23. Orthotropic Conductivity Dialog Box 2330
37.5.24. Anisotropic Conductivity - Principal Components Dialog Box 2331
37.5.25. Anisotropic Conductivity Dialog Box 2331
37.5.26. Species Dialog Box 2332
37.5.27. Reactions Dialog Box 2334
37.5.28. Backward Reaction Parameters Dialog Box 2337
37.5.29. Third-Body Efficiency Dialog Box 2338
37.5.30. Pressure-Dependent Reaction Dialog Box 2339
37.5.31. Coverage-Dependent Reaction Dialog Box 2341
37.5.32. Reference Mass Fractions Dialog Box 2341
37.5.33. Reaction Mechanisms Dialog Box 2342
37.5.34. Site Parameters Dialog Box 2344
37.5.35. Mass Diffusion Coefficients Dialog Box 2344
37.5.36. Thermal Diffusion Coefficients Dialog Box 2346

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.5.37. UDS Diffusion Coefficients Dialog Box 2347
37.5.38. WSGGM User Specified Dialog Box 2348
37.5.39. Gray-Band Absorption Coefficient Dialog Box 2349
37.5.40. Delta-Eddington Scattering Function Dialog Box 2349
37.5.41. Gray-Band Refractive Index Dialog Box 2350
37.5.42. Single Rate Devolatilization Dialog Box 2350
37.5.43. Two Competing Rates Model Dialog Box 2351
37.5.44. CPD Model Dialog Box 2352
37.5.45. Kinetics/Diffusion-Limited Combustion Model Dialog Box 2352
37.5.46. Intrinsic Combustion Model Dialog Box 2353
37.5.47. Multiple Surface Reactions Dialog Box 2354
37.5.48. Edit Material Dialog Box 2355
37.6. Phases 2355
37.6.1. Primary Phase Dialog Box 2357
37.6.2. Secondary Phase Dialog Box 2357
37.6.3. Discrete Phase Dialog Box 2361
37.6.4. Phase Interaction Dialog Box 2363
37.7. Cell Zone Conditions Task Page 2367
37.7.1. Fluid Dialog Box 2369
37.7.2. Solid Dialog Box 2379
37.7.3. Copy Conditions Dialog Box 2381
37.7.4. Operating Conditions Dialog Box 2382
37.7.5. Select Input Parameter Dialog Box 2384
37.7.6. Profiles Dialog Box 2385
37.7.7. Orient Profile Dialog Box 2387
37.7.8. Write Profile Dialog Box 2388
37.8. Boundary Conditions Task Page 2389
37.8.1. Axis Dialog Box 2392
37.8.2. Degassing Dialog Box 2392
37.8.3. Exhaust Fan Dialog Box 2393
37.8.4. Fan Dialog Box 2397
37.8.5. Inlet Vent Dialog Box 2400
37.8.6. Intake Fan Dialog Box 2406
37.8.7. Interface Dialog Box 2411
37.8.8. Interior Dialog Box 2411
37.8.9. Mass-Flow Inlet Dialog Box 2411
37.8.10. Mass Flow Outlet Dialog Box 2417
37.8.11. Outflow Dialog Box 2420
37.8.12. Outlet Vent Dialog Box 2422
37.8.13. Periodic Dialog Box 2427
37.8.14. Porous Jump Dialog Box 2427
37.8.15. Pressure Far-Field Dialog Box 2430
37.8.16. Pressure Inlet Dialog Box 2434
37.8.17. Pressure Outlet Dialog Box 2439
37.8.18. Radiator Dialog Box 2445
37.8.19. RANS/LES Interface Dialog Box 2446
37.8.20. Symmetry Dialog Box 2447
37.8.21. Velocity Inlet Dialog Box 2447
37.8.22. Wall Dialog Box 2456
37.8.23. Periodic Conditions Dialog Box 2468
37.9. Overset Interfaces Task Page 2470
37.9.1. Create/Edit Overset Interfaces Dialog Box 2470

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.10. Dynamic Mesh Task Page 2471
37.10.1. Mesh Method Settings Dialog Box 2474
37.10.2. Mesh Scale Info Dialog Box 2477
37.10.3. Options Dialog Box 2478
37.10.4. In-Cylinder Output Controls Dialog Box 2481
37.10.5. Six DOF Properties Dialog Box 2482
37.10.6. Flow Controls Dialog Box 2485
37.10.7. Dynamic Mesh Events Dialog Box 2486
37.10.8. Define Event Dialog Box 2487
37.10.9. Events Preview Dialog Box 2489
37.10.10. Dynamic Mesh Zones Dialog Box 2490
37.10.11. Orientation Calculator Dialog Box 2497
37.10.12. Inflation Settings Dialog Box 2498
37.10.13. CutCell Boundary Zones Info Dialog Box 2498
37.10.14. Zone Scale Info Dialog Box 2499
37.10.15. Zone Motion Dialog Box 2499
37.10.16. Mesh Motion Dialog Box 2500
37.10.17. Autosave Case During Mesh Motion Preview Dialog Box 2502
37.11. Reference Values Task Page 2502
37.12. Solution Task Page 2504
37.13. Solution Methods Task Page 2504
37.13.1. Relaxation Options Dialog Box 2508
37.14. Solution Controls Task Page 2508
37.14.1. Equations Dialog Box 2511
37.14.2. Solution Limits Dialog Box 2511
37.14.3. Advanced Solution Controls Dialog Box 2513
37.15. Solution Initialization Task Page 2520
37.15.1. Patch Dialog Box 2522
37.15.2. Hybrid Initialization Dialog Box 2524
37.16. Calculation Activities Task Page 2525
37.16.1. Autosave Dialog Box 2527
37.16.2. Data File Quantities Dialog Box 2529
37.16.3. Automatic Export Dialog Box 2530
37.16.4. Automatic Particle History Data Export Dialog Box 2533
37.16.5. Execute Commands Dialog Box 2535
37.16.6. Define Macro Dialog Box 2536
37.16.7. Automatic Solution Initialization and Case Modification Dialog Box 2536
37.17. Run Calculation Task Page 2537
37.17.1. Solution Steering Dialog Box 2541
37.17.2. Case Check Dialog Box 2543
37.17.3. Adaptive Time Step Settings Dialog Box 2544
37.17.4. Variable Time Step Settings Dialog Box 2545
37.17.5. Sampling Options Dialog Box 2546
37.17.6. Acoustic Signals Dialog Box 2547
37.17.7. Acoustic Sources FFT Dialog Box 2549
37.18. Results Task Page 2553
37.19. Graphics and Animations Task Page 2554
37.19.1. Profile Options Dialog Box 2556
37.19.2. Vector Options Dialog Box 2557
37.19.3. Custom Vectors Dialog Box 2558
37.19.4. Vector Definitions Dialog Box 2558
37.19.5. Path Style Attributes Dialog Box 2559

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.19.6. Ribbon Attributes Dialog Box 2560
37.19.7. Particle Filter Attributes 2561
37.19.8. Reporting Variables Dialog Box 2562
37.19.9. Track Style Attributes Dialog Box 2562
37.19.10. Particle Sphere Style Attributes Dialog Box 2563
37.19.11. Particle Vector Style Attributes Dialog Box 2564
37.19.12. Sweep Surface Dialog Box 2565
37.19.13. Create Surface Dialog Box 2566
37.19.14. Animate Dialog Box 2567
37.19.15. Save Picture Dialog Box 2568
37.19.16. Playback Dialog Box 2571
37.19.17. Display Options Dialog Box 2573
37.19.18. Scene Description Dialog Box 2577
37.19.19. Display Properties Dialog Box 2578
37.19.20. Transformations Dialog Box 2580
37.19.21. Iso-Value Dialog Box 2581
37.19.22. Pathline Attributes Dialog Box 2582
37.19.23. Bounding Frame Dialog Box 2582
37.19.24. Views Dialog Box 2583
37.19.25. Write Views Dialog Box 2584
37.19.26. Mirror Planes Dialog Box 2585
37.19.27. Graphics Periodicity Dialog Box 2586
37.19.28. Camera Parameters Dialog Box 2587
37.19.29. Lights Dialog Box 2588
37.19.30. Colormap Dialog Box 2589
37.19.31. Colormap Editor Dialog Box 2591
37.19.32. Annotate Dialog Box 2592
37.20. Plots Task Page 2593
37.20.1. Solution XY Plot Dialog Box 2595
37.20.2. Histogram Dialog Box 2598
37.20.3. File XY Plot Dialog Box 2599
37.20.4. Plot Profile Data Dialog Box 2601
37.20.5. Plot Interpolated Data Dialog Box 2601
37.20.6. Fourier Transform Dialog Box 2602
37.20.7. Plot/Modify Input Signal Dialog Box 2605
37.20.8. Axes Dialog Box 2608
37.20.9. Curves Dialog Box 2610
37.21. Reports Task Page 2611
37.21.1. Flux Reports Dialog Box 2613
37.21.2. Force Reports Dialog Box 2614
37.21.3. Projected Surface Areas Dialog Box 2615
37.21.4. Surface Integrals Dialog Box 2616
37.21.5. Volume Integrals Dialog Box 2620
37.21.6. Sample Trajectories Dialog Box 2622
37.21.7. Trajectory Sample Histograms Dialog Box 2623
37.21.8. Particle Summary Dialog Box 2625
37.21.9. Heat Exchanger Report Dialog Box 2625
37.21.10. Parameters Dialog Box 2627
37.21.11. Use Input Parameter in Scheme Procedure Dialog Box 2628
37.21.12. Use Input Parameter for UDF Dialog Box 2629
37.21.13. Rename Dialog Box 2630
37.21.14. Input Parameter Properties Dialog Box 2631

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.21.15. Save Output Parameter Dialog Box 2631
37.22. Parameters and Customization Task Page 2632
38. Ribbon Reference Guide 2633
38.1. File Ribbon Tab 2633
38.1.1. File/Read/Mesh... 2634
38.1.1.1. Read Mesh Options Dialog Box 2634
38.1.2. File/Read/Case... 2635
38.1.3. File/Read/Data... 2636
38.1.4. File/Read/Case & Data... 2636
38.1.5. File/Read/PDF... 2636
38.1.6. File/Read/ISAT Table... 2636
38.1.7. File/Read/DTRM Rays... 2636
38.1.8. File/Read/View Factors... 2636
38.1.9. File/Read/Profile... 2636
38.1.10. File/Read/Scheme... 2637
38.1.11. File/Read/Journal... 2637
38.1.12. File/Write/Case... 2637
38.1.13. File/Write/Data... 2637
38.1.14. File/Write/Case & Data... 2637
38.1.15. File/Write/PDF... 2638
38.1.16. File/Write/ISAT Table... 2638
38.1.17. File/Write/Flamelet... 2638
38.1.18. File/Write/Surface Clusters... 2638
38.1.19. File/Write/Profile... 2638
38.1.20. File/Write/Autosave... 2638
38.1.21. File/Write/Boundary Mesh... 2638
38.1.22. File/Write/Start Journal... 2638
38.1.23. File/Write/Stop Journal 2639
38.1.24. File/Write/Start Transcript... 2639
38.1.25. File/Write/Stop Transcript 2639
38.1.26. File/Import/ABAQUS/Input File... 2639
38.1.27. File/Import/ABAQUS/Filbin File... 2639
38.1.28. File/Import/ABAQUS/ODB File... 2639
38.1.29. File/Import/CFX/Definition File... 2639
38.1.30. File/Import/CFX/Result File... 2639
38.1.31. File/Import/CGNS/Mesh... 2639
38.1.32. File/Import/CGNS/Data... 2640
38.1.33. File/Import/CGNS/Mesh & Data... 2640
38.1.34. File/Import/EnSight... 2640
38.1.35. File/Import/FIDAP... 2640
38.1.36. File/Import/GAMBIT... 2640
38.1.37. File/Import/HYPERMESH ASCII... 2640
38.1.38. File/Import/I-deas Universal... 2640
38.1.39. File/Import/LSTC/Input File... 2640
38.1.40. File/Import/LSTC/State File... 2640
38.1.41. File/Import/Marc POST... 2640
38.1.42. File/Import/Mechanical APDL/Input File... 2641
38.1.43. File/Import/Mechanical APDL/Result File... 2641
38.1.44. File/Import/NASTRAN/Bulkdata File... 2641
38.1.45. File/Import/NASTRAN/Op2 File... 2641
38.1.46. File/Import/PATRAN/Neutral File... 2641
38.1.47. File/Import/PLOT3D/Grid File... 2641

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.1.48. File/Import/PLOT3D/Result File... 2641
38.1.49. File/Import/PTC Mechanica Design... 2641
38.1.50. File/Import/Tecplot... 2641
38.1.51. File/Import/Fluent 4 Case File... 2642
38.1.52. File/Import/PreBFC File... 2642
38.1.53. File/Import/Partition/Metis... 2642
38.1.54. File/Import/Partition/Metis Zone... 2642
38.1.55. File/Import/CHEMKIN Mechanism... 2642
38.1.55.1. CHEMKIN Mechanism Import Dialog Box 2642
38.1.56. File/Export/Solution Data... 2644
38.1.56.1. Export Dialog Box 2644
38.1.57. File/Export/Particle History Data... 2648
38.1.57.1. Export Particle History Data Dialog Box 2648
38.1.58. File/Export/During Calculation/Solution Data... 2650
38.1.59. File/Export/During Calculation/Particle History Data... 2650
38.1.60. File/Export to CFD-Post... 2650
38.1.60.1. Export to CFD-Post Dialog Box 2650
38.1.61. File/Solution Files... 2651
38.1.61.1. Solution Files Dialog Box 2651
38.1.62. File/Interpolate... 2652
38.1.62.1. Interpolate Data Dialog Box 2652
38.1.63. File/FSI Mapping/Volume... 2653
38.1.63.1. Volume FSI Mapping Dialog Box 2653
38.1.64. File/FSI Mapping/Surface... 2656
38.1.64.1. Surface FSI Mapping Dialog Box 2656
38.1.65. File/Save Picture... 2658
38.1.66. File/Data File Quantities... 2658
38.1.67. File/Batch Options... 2659
38.1.67.1. Batch Options Dialog Box 2659
38.1.68. File/Exit 2659
38.2. Dialog Boxes Available from the Ribbon 2660
38.2.1. 1D Simulation Library Dialog Box 2662
38.2.2. Activate Cell Zones Dialog Box 2662
38.2.3. Adaption Display Options Dialog Box 2663
38.2.4. Adjacency Dialog Box 2664
38.2.5. Animation Definition Dialog Box 2666
38.2.6. Animation Recording Options Dialog Box 2668
38.2.7. Anisotropic Adaption Dialog Box 2671
38.2.8. Auto Create Options Dialog Box 2672
38.2.9. Auto Partition Mesh Dialog Box 2672
38.2.10. Boundary Adaption Dialog Box 2673
38.2.11. Compiled UDFs Dialog Box 2675
38.2.12. Contours Dialog Box 2676
38.2.13. Convergence Conditions Dialog Box 2678
38.2.14. Create/Edit Mesh Interfaces Dialog Box 2680
38.2.15. Custom Field Function Calculator Dialog Box 2683
38.2.16. Custom Laws Dialog Box 2684
38.2.17. Deactivate Cell Zones Dialog Box 2685
38.2.18. Define Control Points Dialog Box 2685
38.2.19. Delete Cell Zones Dialog Box 2687
38.2.20. DPM Report Definition Dialog Box 2687
38.2.21. DPM Source Report Definition 2690

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.2.22. Drag Report Definition Dialog Box 2691
38.2.23. DTRM Graphics Dialog Box 2694
38.2.24. DTRM Rays Dialog Box 2695
38.2.25. Edit Mesh Interfaces Dialog Box 2696
38.2.26. Edit Report File Dialog Box 2698
38.2.27. Edit Report Plot Dialog Box 2700
38.2.28. Execute on Demand Dialog Box 2702
38.2.29. Expression Report Definition Dialog Box 2703
38.2.30. Field Function Definitions Dialog Box 2704
38.2.31. Flux Report Definition Dialog Box 2705
38.2.32. Fuse Face Zones Dialog Box 2708
38.2.33. Geometry Based Adaption Controls Dialog Box 2709
38.2.34. Geometry Based Adaption Dialog Box 2709
38.2.35. Gradient Adaption Dialog Box 2710
38.2.36. Import Particle Data Dialog Box 2713
38.2.37. Imprint Surface Dialog Box 2714
38.2.38. Improve Mesh Dialog Box 2715
38.2.39. Injections Dialog Box 2716
38.2.40. Input Summary Dialog Box 2717
38.2.41. Interface Options Dialog Box 2717
38.2.42. Interpreted UDFs Dialog Box 2719
38.2.43. Iso-Clip Dialog Box 2720
38.2.44. Iso-Surface Dialog Box 2721
38.2.45. Iso-Value Adaption Dialog Box 2722
38.2.46. Lift Report Definition Dialog Box 2724
38.2.47. Line/Rake Surface Dialog Box 2727
38.2.48. Manage Adaption Registers Dialog Box 2728
38.2.49. Merge Zones Dialog Box 2730
38.2.50. Mesh Adaption Controls Dialog Box 2730
38.2.51. Mesh Interfaces Dialog Box 2732
38.2.52. Mesh Morpher/Optimizer Dialog Box 2734
38.2.53. MiniVAS Settings Dialog Box 2742
38.2.54. Mixing Planes Dialog Box 2744
38.2.55. Moment Report Definition Dialog Box 2746
38.2.56. Motion Settings Dialog Box 2749
38.2.57. New Report File Dialog Box 2754
38.2.58. New Report Plot Dialog Box 2755
38.2.59. Objective Function Definition Dialog Box 2757
38.2.60. Optimization History Monitor Dialog Box 2759
38.2.61. Parallel Connectivity Dialog Box 2760
38.2.62. Parameter Bounds Dialog Box 2760
38.2.63. Particle Tracks Dialog Box 2761
38.2.64. Partition Surface Dialog Box 2766
38.2.65. Partitioning and Load Balancing Dialog Box 2767
38.2.66. Pathlines Dialog Box 2771
38.2.67. Picture Options Dialog Box 2776
38.2.68. Plane Surface Dialog Box 2777
38.2.69. Point Surface Dialog Box 2780
38.2.70. Quadric Surface Dialog Box 2781
38.2.71. Region Adaption Dialog Box 2782
38.2.72. Replace Cell Zone Dialog Box 2785
38.2.73. Report Definitions Dialog Box 2785

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.2.74. Report File Definitions Dialog Box 2787
38.2.75. Report Plot Definitions Dialog Box 2788
38.2.76. Residual Monitors Dialog Box 2789
38.2.77. Rotate Mesh Dialog Box 2792
38.2.78. S2S Information Dialog Box 2792
38.2.79. Separate Cell Zones Dialog Box 2793
38.2.80. Separate Face Zones Dialog Box 2794
38.2.81. Set Injection Properties Dialog Box 2796
38.2.82. Set Multiple Injection Properties Dialog Box 2803
38.2.83. Shell Conduction Layers Dialog Box 2804
38.2.84. Shell Conduction Manager Dialog Box 2806
38.2.85. Surface Meshes Dialog Box 2807
38.2.86. Surface Report Definition Dialog Box 2807
38.2.87. Surfaces Dialog Box 2810
38.2.88. Thread Control Dialog Box 2811
38.2.89. Transform Surface Dialog Box 2812
38.2.90. Translate Mesh Dialog Box 2813
38.2.91. Turbo 2D Contours Dialog Box 2814
38.2.92. Turbo Averaged Contours Dialog Box 2816
38.2.93. Turbo Averaged XY Plot Dialog Box 2817
38.2.94. Turbo Options Dialog Box 2819
38.2.95. Turbo Report Dialog Box 2819
38.2.96. Turbo Topology Dialog Box 2822
38.2.97. UDF Library Manager Dialog Box 2823
38.2.98. User-Defined Fan Model Dialog Box 2824
38.2.99. User-Defined Function Hooks Dialog Box 2825
38.2.100. User-Defined Memory Dialog Box 2827
38.2.101. User Defined Report Definition Dialog Box 2828
38.2.102. User-Defined Scalars Dialog Box 2829
38.2.103. Vectors Dialog Box 2831
38.2.104. Video Control Dialog Box 2833
38.2.105. V-LAN Settings Dialog Box 2836
38.2.106. Volume Adaption Dialog Box 2838
38.2.107. Volume Report Definition Dialog Box 2839
38.2.108. Warning Dialog Box 2841
38.2.109. Yplus/Ystar Adaption Dialog Box 2842
38.2.110. Zone Surface Dialog Box 2844
A. ANSYS Fluent Model Compatibility 2847
B. ANSYS Fluent File Formats 2851
B.1. Case and Data File Formats 2851
B.1.1. Guidelines 2851
B.1.2. Formatting Conventions in Binary and Formatted Files 2851
B.1.3. Grid Sections 2852
B.1.3.1. Comment 2852
B.1.3.2. Header 2853
B.1.3.3. Dimensions 2853
B.1.3.4. Nodes 2853
B.1.3.5. Periodic Shadow Faces 2854
B.1.3.6. Cells 2855
B.1.3.7. Faces 2856
B.1.3.8. Face Tree 2858
B.1.3.9. Cell Tree 2858

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

B.1.3.10. Interface Face Parents 2859
B.1.3.11. Example Files 2860
B.1.3.11.1. Example 1 2860
B.1.3.11.2. Example 2 2861
B.1.3.11.3. Example 3 2861
B.1.4. Other (Non-Grid) Case Sections 2863
B.1.4.1. Zone 2863
B.1.4.2. Partitions 2865
B.1.5. Data Sections 2865
B.1.5.1. Grid Size 2865
B.1.5.2. Data Field 2866
B.1.5.3. Residuals 2867
B.2. Mesh Morpher/Optimizer File Formats 2867
B.3. Shell Conduction Settings File Format 2868
B.4. 3D Fan Curve File Format 2869
C. Nomenclature 2871
D. Controlling CHEMKIN-CFD Solver Parameters Using Text Commands 2875
D.1. Advanced Parameters Used in the Steady-State Solution Algorithm 2879
D.2. Setting Up Monitor Cells for the ANSYS CHEMKIN-CFD Chemistry Solver 2880
D.3. Diagnostic Files and Error Messages 2880
D.4. Error Messages Printed in the ANSYS Fluent Graphical User Interface 2882
D.5. Diagnostic Messages in the `KINetics-log.txt` File 2884
Bibliography 2887
Index 2895

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Using This Manual

This preface is divided into the following sections:

1. The Contents of This Manual
2. The Contents of the Fluent Manuals
3. Typographical Conventions
4. Mathematical Conventions
5. Technical Support

1. The Contents of This Manual

The ANSYS Fluent User's Guide tells you what you need to know to use ANSYS Fluent.

Important

Under U.S. and international copyright law, ANSYS, Inc. is unable to distribute copies of the papers listed in the bibliography, other than those published internally by ANSYS, Inc. Use your library or a document delivery service to obtain copies of copyrighted papers.

A brief description of what is in each chapter follows:

- Graphical User Interface (GUI) (p. 1), describes the mechanics of using the graphical user interface and the GUI on-line help.
- Text User Interface (TUI) (p. 27), describes the mechanics of using the text interface and the TUI on-line help. (See the Text Command List for information about specific text interface commands.)
- Reading and Writing Files (p. 39), describes the files that ANSYS Fluent can read and write, including picture files.
- Unit Systems (p. 111), describes how to use the standard and custom unit systems available in ANSYS Fluent.
- Reading and Manipulating Meshes (p. 115), describes the various sources of computational meshes and explains how to obtain diagnostic information about the mesh and how to modify it by scaling, translating, and other methods. This chapter also contains information about the use of non-conformal meshes.
- Cell Zone and Boundary Conditions (p. 237), describes the different types of boundary conditions available in ANSYS Fluent, when to use them, how to define them, and how to define boundary profiles and volumetric sources and fix the value of a variable in a particular region. It also contains information about porous media and lumped parameter models.
- Physical Properties (p. 459), describes the physical properties of materials and the equations that ANSYS Fluent uses to compute the properties from the information that you input.
- Modeling Basic Fluid Flow (p. 573), describes the governing equations and physical models used by ANSYS Fluent to compute fluid flow (including periodic flow, swirling and rotating flows, compressible flows, and inviscid flows), as well as the inputs you need to provide to use these models.
- Modeling Flows with Moving Reference Frames (p. 603), describes the use of single moving reference frames, multiple moving reference frames, and mixing planes in ANSYS Fluent.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Modeling Flows Using Sliding and Dynamic Meshes (p. 627), describes the use of sliding and deforming meshes in ANSYS Fluent.
- Modeling Turbulence (p. 749), describes the use of the turbulent flow models in ANSYS Fluent.
- Modeling Heat Transfer (p. 825), describes the use of the physical models in ANSYS Fluent to compute heat transfer (including convective and conductive heat transfer, natural convection, radiative heat transfer, and periodic heat transfer), as well as the inputs you need to provide to use these models.
- Modeling Heat Exchangers (p. 919), describes the use of the heat exchanger models in ANSYS Fluent.
- Modeling Species Transport and Finite-Rate Chemistry (p. 957), describes the use of the finite-rate chemistry models in ANSYS Fluent. This chapter also provides information about modeling species transport in non-reacting flows.
- Modeling Non-Premixed Combustion (p. 1025), describes the use of the non-premixed combustion model in ANSYS Fluent. This chapter includes details about using prePDF.
- Modeling Premixed Combustion (p. 1089), describes the use of the premixed combustion model in ANSYS Fluent.
- Modeling Partially Premixed Combustion (p. 1099), describes the use of the partially premixed combustion model in ANSYS Fluent.
- Modeling a Composition PDF Transport Problem (p. 1113), describes the use of the composition PDF transport model in ANSYS Fluent.
- Using Chemistry Acceleration (p. 1127), describes the use of methods to accelerate computations for detailed chemical mechanisms involving laminar and turbulent flames.
- Modeling Engine Ignition (p. 1139), describes the use of the engine ignition models in ANSYS Fluent.
- Modeling Pollutant Formation (p. 1153), describes the use of the models for the formation of NOx, SOx, and soot in ANSYS Fluent.
- Predicting Aerodynamically Generated Noise (p. 1205), describes the use of the acoustics model in ANSYS Fluent.
- Modeling Discrete Phase (p. 1235), describes the use of the discrete phase models in ANSYS Fluent.
- Modeling Multiphase Flows (p. 1365), describes the use of the general multiphase models in ANSYS Fluent (VOF, mixture, and Eulerian).
- Modeling Solidification and Melting (p. 1527), describes the use of the solidification and melting model in ANSYS Fluent.
- Modeling Eulerian Wall Films (p. 1535), describes the use of the Eulerian wall film model in ANSYS Fluent.
- Modeling Electric Potential Field (p. 1547), describes the use of the electric potential model in ANSYS Fluent.
- Using the Solver (p. 1551), describes the use of the ANSYS Fluent solvers.
- Adapting the Mesh (p. 1681), describes the use of the solution-adaptive mesh refinement feature in ANSYS Fluent.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Creating Surfaces and Cell Registers for Displaying and Reporting Data (p. 1709), describes how to create surfaces in the domain on which you can examine ANSYS Fluent solution data.
- Displaying Graphics (p. 1747), describes the use of the graphics tools to examine your ANSYS Fluent solution.
- Reporting Alphanumeric Data (p. 1885), describes how to obtain reports of fluxes, forces, surface integrals, and other solution data.
- Field Function Definitions (p. 1931), describes the flow variables that appear in the variable selection drop-down lists in ANSYS Fluent dialog boxes, and tells you how to create your own custom field functions.
- Parallel Processing (p. 2009), describes the use of the parallel processing features in ANSYS Fluent. This chapter also provides information about partitioning your mesh for parallel processing.
- Design Analysis and Optimization (p. 2067), describes the use of the adjoint solver and the mesh morpher/optimizer. These features allow you to modify the design and solve optimization problems in ANSYS Fluent.
- Task Page Reference Guide (p. 2159), describes the use of the task pages within the ANSYS Fluent user interface.
- Ribbon Reference Guide (p. 2633), describes the **File** ribbon tab and other dialog boxes that are available from the ribbon at the top of the ANSYS Fluent user interface.
- Appendix A: ANSYS Fluent Model Compatibility (p. 2847), presents a series of tables outlining the compatibility of several ANSYS Fluent model categories.
- Appendix B: ANSYS Fluent File Formats (p. 2851), presents information about the contents and formats of ANSYS Fluent case, data, and mesh morpher/optimizer files.
- Appendix C: Nomenclature (p. 2871), presents a brief listing of the physical properties and fluid dynamic quantities used throughout the ANSYS Fluent documentation.

2. The Contents of the Fluent Manuals

The manuals listed below form the Fluent product documentation set. They include descriptions of the procedures, commands, and theoretical details needed to use Fluent products.

- Fluent Getting Started Guide contains general information about getting started with using Fluent and provides details about starting, running, and exiting the program.
- Fluent Migration Manual contains information about transitioning from the previous release of Fluent, including details about new features, output changes, and text command list changes.
- Fluent User's Guide contains detailed information about running a simulation using the solution mode of Fluent, including information about the user interface, reading and writing files, defining boundary conditions, setting up physical models, calculating a solution, analyzing your results, and revising the design.
- ANSYS Fluent Meshing Migration Manual contains information about transitioning from the previous release of Fluent Meshing, including descriptions of new features and text command list changes.
- ANSYS Fluent Meshing User's Guide contains detailed information about creating 3D meshes using the meshing mode of Fluent.

  Related video help on meshing can be found on the ANSYS How To Videos page.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Fluent in Workbench User's Guide contains information about getting started with and using Fluent within the Workbench environment.
- Fluent Theory Guide contains reference information for how the physical models are implemented in Fluent.
- Fluent Customization Manual contains information about writing / using user-defined functions (UDFs) and customizing the Fluent graphical user interface (GUI) for easy management of the data used by UDFs.
- Fluent Tutorial Guide contains a number of examples of various flow problems with detailed instructions, commentary, and postprocessing of results.

  The latest updates of the ANSYS Fluent tutorials are available on the ANSYS Customer Portal. To access tutorials and their input files on the ANSYS Customer Portal, go to http://support.ansys.com/training.
- The latest updates of the ANSYS Fluent Meshing tutorials are available on the ANSYS Customer Portal. To access tutorials and their input files on the ANSYS Customer Portal, go to http://support.ansys.com/training.
- Fluent Text Command List contains a brief description of each of the commands in Fluent's solution mode text interface.
- ANSYS Fluent Meshing Text Command List contains a brief description of each of the commands in Fluent's meshing mode text interface.
- ANSYS Fluent Advanced Add-On Modules contains information about the usage of the different advanced Fluent add-on modules, which are applicable for specific modeling needs.
  - Part I: ANSYS Fluent Battery Module contains information about the background and usage of Fluent's Battery Module that allows you to analyze the behavior of electric batteries.
  - Part II: ANSYS Fluent Continuous Fiber Module contains information about the background and usage of Fluent's Continuous Fiber Module that allows you to analyze the behavior of fiber flow, fiber properties, and coupling between fibers and the surrounding fluid due to the strong interaction that exists between the fibers and the surrounding gas.
  - Part III: ANSYS Fluent Macroscopic Particle Module contains information about the background and usage of Fluent's Macroscopic Particle Model (MPM) that predicts the behavior of large (macroscopic) particles and their interaction with the fluid flow, walls, and other particles.
  - Part IV: ANSYS Fluent Fuel Cell Modules contains information about the background and the usage of two separate add-on fuel cell models for Fluent that allow you to model polymer electrolyte membrane fuel cells (PEMFC), solid oxide fuel cells (SOFC), and electrolysis with Fluent.
  - Part V: ANSYS Fluent Magnetohydrodynamics (MHD) Module contains information about the background and usage of Fluent's Magnetohydrodynamics (MHD) Module that allows you to analyze the behavior of electrically conducting fluid flow under the influence of constant (DC) or oscillating (AC) electromagnetic fields.
  - Part VI: ANSYS Fluent Population Balance Module contains information about the background and usage of Fluent's Population Balance Module that allows you to analyze multiphase flows involving size distributions where particle population (as well as momentum, mass, and energy) require a balance equation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Fluent as a Server User's Guide contains information about the usage of Fluent as a Server which allows you to connect to a Fluent session and issue commands from a remote client application.
- Running ANSYS Fluent Using a Load Manager contains information about using third-party load managers with ANSYS Fluent.
  - Part I: Running ANSYS Fluent Under LSF contains information about using Fluent with Platform Computing's LSF software, a distributed computing resource management tool.
  - Part II: Running ANSYS Fluent Under PBS Professional contains information about using Fluent with Altair PBS Professional, an open workload management tool for local and distributed environments.
  - Part III: Running ANSYS Fluent Under SGE contains information about using Fluent with Univa Grid Engine (formerly Sun Grid Engine) software, a distributed computing resource management tool.

Related video help for Fluent can be found on the ANSYS How To Videos page.

3. Typographical Conventions

Several typographical conventions are used in this manual's text to help you find commands in the user interface.

- Different type styles are used to indicate graphical user interface items and text interface items. For example:

  Iso-Surface dialog box
  `surface/iso-surface` text command

- The text interface type style is also used when illustrating exactly what appears on the screen to distinguish it from the narrative text. In this context, user inputs are typically shown in boldface. For example,

```
solve/initialize/set-fmg-initialization

 Customize your FMG initialization:
   set the number of multigrid levels [5]

   set FMG parameters on levels ..

    residual reduction on level 1 is:  [0.001]
    number of cycles on level 1 is:  [10]  100

    residual reduction on level 2 is:  [0.001]
    number of cycles on level 2 is:  [50]  100
```

- Mini flow charts are used to guide you through the ribbon or the tree, leading you to a specific option, dialog box, or task page. The following tables list the meaning of each symbol in the mini flow charts.

Table 1: Mini Flow Chart Symbol Descriptions

Symbol	Indicated Action
	Look at the ribbon
	Look at the tree
	Double-click to open task page
	Select from task page

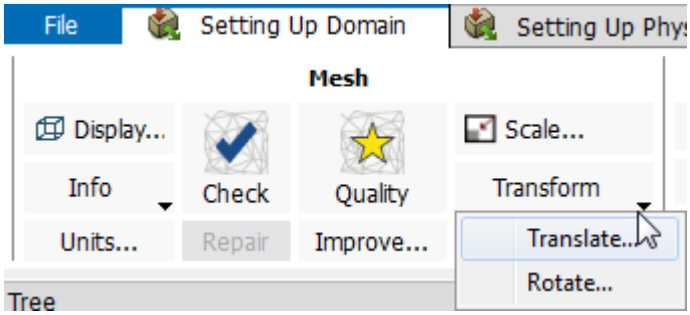
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Right-click the preceding item

For example,

Setting Up Domain → Mesh → Transform → Translate...

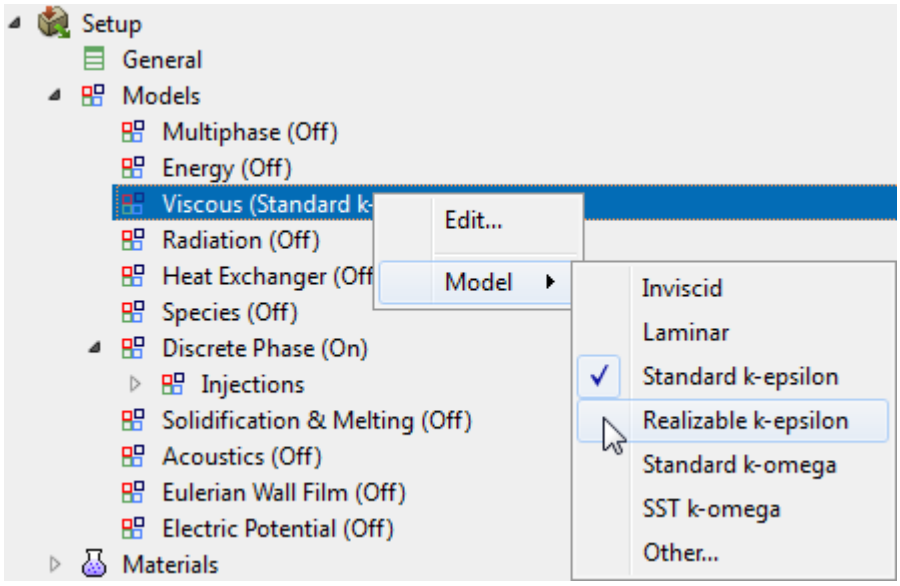
indicates selecting the **Setting Up Domain** ribbon tab, clicking **Transform** (in the **Mesh** group box) and selecting **Translate...**, as indicated in the figure below:



And

Setup → Models → Viscous Model → Realizable k-epsilon

indicates expanding the **Setup** and **Models** branches, right-clicking **Viscous**, and selecting **Realizable k-epsilon** from the **Model** sub-menu, as shown in the following figure:

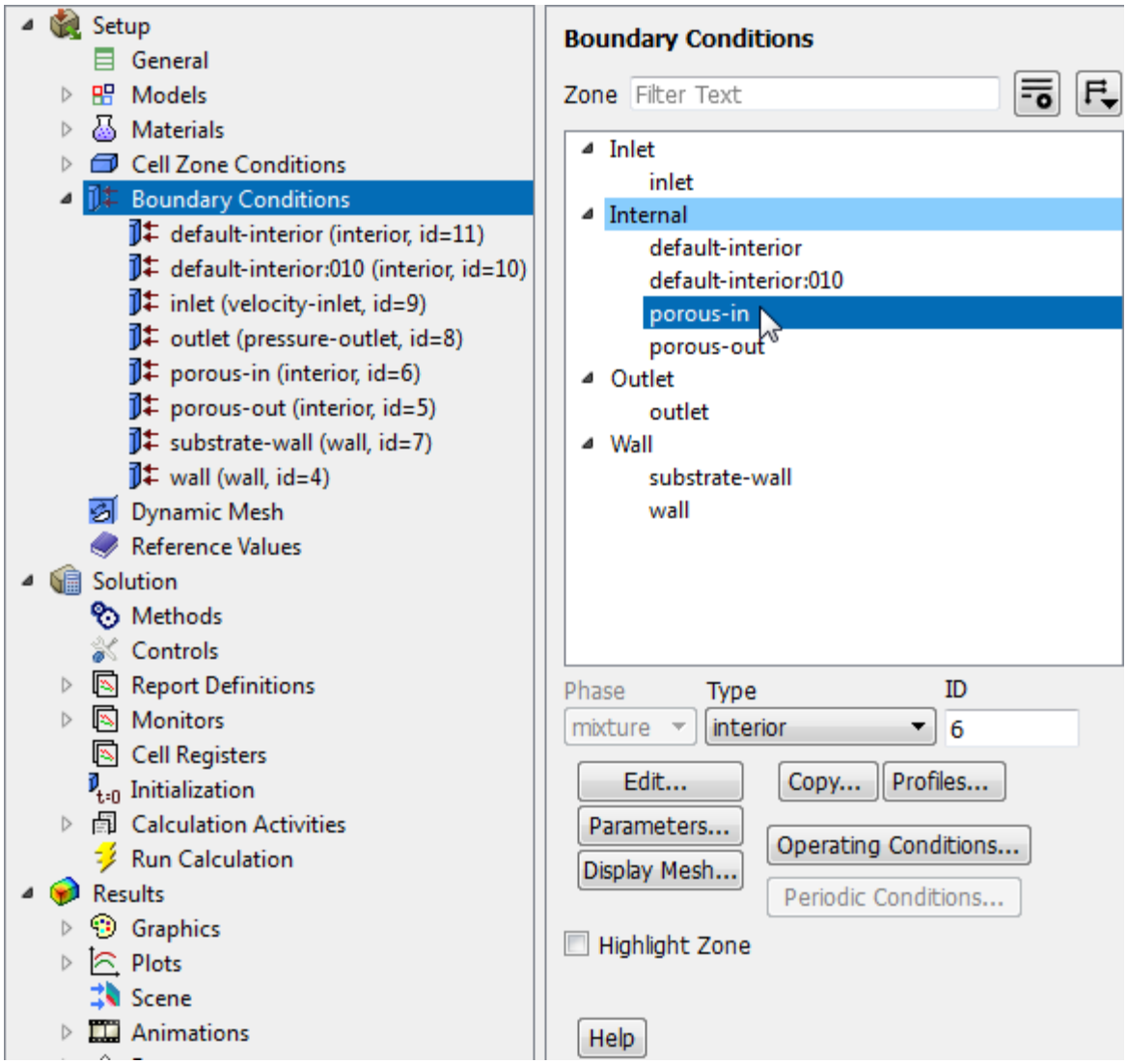


And

Setup → Boundary Conditions → porous-in

indicates opening the task page as shown below:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In this manual, mini flow charts usually accompany a description of a dialog box or command, or a screen illustration showing how to use the dialog box or command. They show you how to quickly access a command or dialog box without having to search the surrounding material.

- In-text references to **File** ribbon tab selections can be indicated using a "/". For example **File/Write/Case...** indicates clicking the **File** ribbon tab and selecting **Case...** from the **Write** submenu (which opens the **Select File** dialog box).

4. Mathematical Conventions

- Where possible, vector quantities are displayed with a raised arrow (e.g., $\vec{a}$, $\vec{A}$). Boldfaced characters are reserved for vectors and matrices as they apply to linear algebra (e.g., the identity matrix, $\mathbf{I}$).

- The operator ∇, referred to as grad, nabla, or del, represents the partial derivative of a quantity with respect to all directions in the chosen coordinate system. In Cartesian coordinates, ∇ is defined to be

$$\frac{\partial}{\partial x}\vec{i}+\frac{\partial}{\partial y}\vec{j}+\frac{\partial}{\partial z}\vec{k} \tag{1}$$

∇ appears in several ways:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– The gradient of a scalar quantity is the vector whose components are the partial derivatives; for example,

$$\nabla p=\frac{\partial p}{\partial x}\vec{i}+\frac{\partial p}{\partial y}\vec{j}+\frac{\partial p}{\partial z}\vec{k} \tag{2}$$

– The gradient of a vector quantity is a second-order tensor; for example, in Cartesian coordinates,

$$\nabla(\vec{v})=\left(\frac{\partial}{\partial x}\vec{i}+\frac{\partial}{\partial y}\vec{j}+\frac{\partial}{\partial z}\vec{k}\right)\left(v_x\vec{i}+v_j\vec{j}+v_z\vec{k}\right) \tag{3}$$

This tensor is usually written as

$$\begin{pmatrix} \frac{\partial v_x}{\partial x} & \frac{\partial v_x}{\partial y} & \frac{\partial v_x}{\partial z} \\ \frac{\partial v_y}{\partial x} & \frac{\partial v_y}{\partial y} & \frac{\partial v_y}{\partial z} \\ \frac{\partial v_z}{\partial x} & \frac{\partial v_z}{\partial y} & \frac{\partial v_z}{\partial z} \end{pmatrix} \tag{4}$$

– The divergence of a vector quantity, which is the inner product between ∇ and a vector; for example,

$$\nabla\cdot\vec{v}=\frac{\partial v_x}{\partial x}+\frac{\partial v_y}{\partial y}+\frac{\partial v_z}{\partial z} \tag{5}$$

– The operator $\nabla\cdot\nabla$, which is usually written as ∇^2 and is known as the Laplacian; for example,

$$\nabla^2 T=\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}+\frac{\partial^2 T}{\partial z^2} \tag{6}$$

$\nabla^2 T$ is different from the expression $\left(\nabla T\right)^2$, which is defined as

$$\left(\nabla T\right)^2=\left(\frac{\partial T}{\partial x}\right)^2+\left(\frac{\partial T}{\partial y}\right)^2+\left(\frac{\partial T}{\partial z}\right)^2 \tag{7}$$

- An exception to the use of ∇ is found in the discussion of Reynolds stresses in Turbulence in the *Fluent Theory Guide*, where convention dictates the use of Cartesian tensor notation. In this chapter, you will also find that some velocity vector components are written as u, v, and w instead of the conventional v with directional subscripts.

5. Technical Support

If you encounter difficulties while using ANSYS Fluent, please first refer to the section(s) of the manual containing information on the commands you are trying to use or the type of problem you are trying to solve. The product documentation is available from the online help, or from the ANSYS Customer Portal. To access documentation files on the ANSYS Customer Portal, go to http://support.ansys.com/documentation.

If you encounter an error, please write down the exact error message that appeared and note as much information as you can about what you were doing in ANSYS Fluent.

Technical Support for ANSYS, Inc. products is provided either by ANSYS, Inc. directly or by one of our certified ANSYS Support Providers. Please check with the ANSYS Support Coordinator (ASC) at your

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

company to determine who provides support for your company, or go to www.ansys.com and select **Contacts> Contacts and Locations**.

If your support is provided by ANSYS, Inc. directly, Technical Support can be accessed quickly and efficiently from the ANSYS Customer Portal, which is available from the ANSYS Website (www.ansys.com) under **Support > Customer Portal**. The direct URL is: support.ansys.com.

One of the many useful features of the Customer Portal is the Knowledge Resources Search, which can be found on the Home page of the Customer Portal. To use this feature, enter relevant text (error message, etc.) in the Knowledge Resources Search box and click the magnifying glass icon. These Knowledge Resources provide solutions and guidance on how to resolve installation and licensing issues quickly.

NORTH AMERICA

All ANSYS Products except Esterel, Apache and Reaction Design products

Web: Go to the ANSYS Customer Portal (http://support.ansys.com) and select the appropriate option.

Toll-Free Telephone: 1.800.711.7199 (Please have your Customer or Contact ID ready.)

Support for University customers is provided only through the ANSYS Customer Portal.

GERMANY

ANSYS Mechanical Products

Telephone: +49 (0) 8092 7005-55 (CADFEM)

Email: support@cadfem.de

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://support.ansys.com) and select the appropriate option.

National Toll-Free Telephone: (Please have your Customer or Contact ID ready.)

German language: 0800 181 8499

English language: 0800 181 1565

Austria: 0800 297 835

Switzerland: 0800 564 318

International Telephone: (Please have your Customer or Contact ID ready.)

German language: +49 6151 152 9981

English language: +49 6151 152 9982

Email: support-germany@ansys.com

UNITED KINGDOM

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://support.ansys.com) and select the appropriate option.

Telephone: Please have your Customer or Contact ID ready.

UK: 0800 048 0462

Republic of Ireland: 1800 065 6642

Outside UK: +44 1235 420130

Email: support-uk@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

JAPAN

CFX and Mechanical Products

Telephone: +81-3-5324-7305

Email:

Mechanical: japan-ansys-support@ansys.com

Fluent: japan-fluent-support@ansys.com;

CFX: japan-cfx-support@ansys.com;

Polyflow: japan-polyflow-support@ansys.com;

Icepak

Telephone: +81-3-5324-7444

Email: japan-icepak-support@ansys.com

Licensing and Installation

Email: japan-license-support@ansys.com

INDIA

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://support.ansys.com) and select the appropriate option.

Telephone: +91 1 800 209 3475 (toll free) or +91 20 6654 3000 (toll) (Please have your Customer or Contact ID ready.)

FRANCE

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://support.ansys.com) and select the appropriate option.

Toll-Free Telephone: +33 (0) 800 919 225 **Toll Number:** +33 (0) 170 489 087 (Please have your Customer or Contact ID ready.)

Email: support-france@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

BELGIUM

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://support.ansys.com) and select the appropriate option.

Toll-Free Telephone: (0) 800 777 83 **Toll Number:** +32 2 620 0152 (Please have your Customer or Contact ID ready.)

Email: support-benelux@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

SWEDEN

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://support.ansys.com) and select the appropriate option.

Telephone: +46 (0) 10 516 49 00

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Email: support-sweden@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

SPAIN and PORTUGAL

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://support.ansys.com) and select the appropriate option.

Spain: Toll-Free Telephone: 900 933 407 **Toll Number:** +34 9178 78350 (Please have your Customer or Contact ID ready.)

Portugal: Toll-Free Telephone: 800 880 513 (Portugal)

Email: support-spain@ansys.com, support-portugal@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

ITALY

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://support.ansys.com) and select the appropriate option.

Toll-Free Telephone: 800 789 531 **Toll Number:** +39 02 00621386 (Please have your Customer or Contact ID ready.)

Email: support-italy@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

TAIWAN, REPUBLIC OF CHINA

Telephone: 866 22725 5828

K0REA

Telephone: 82-2-3441-5000

CHINA

Toll-Free Telephone: 400 819 8999 **Toll Number:** +86 10 82861715

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 1: Graphical User Interface (GUI)

The Fluent graphical interface consists of a ribbon (tabs), a toolbar, a tree, a task page, a graphics toolbar, graphics windows, and a console, which is a textual command line interface (described in Text User Interface (TUI) (p. 27)). You will also have access to the dialog boxes via the ribbon or task pages.

1.1. GUI Components
1.2. Customizing the Graphical User Interface
1.3. Using the GUI Help System

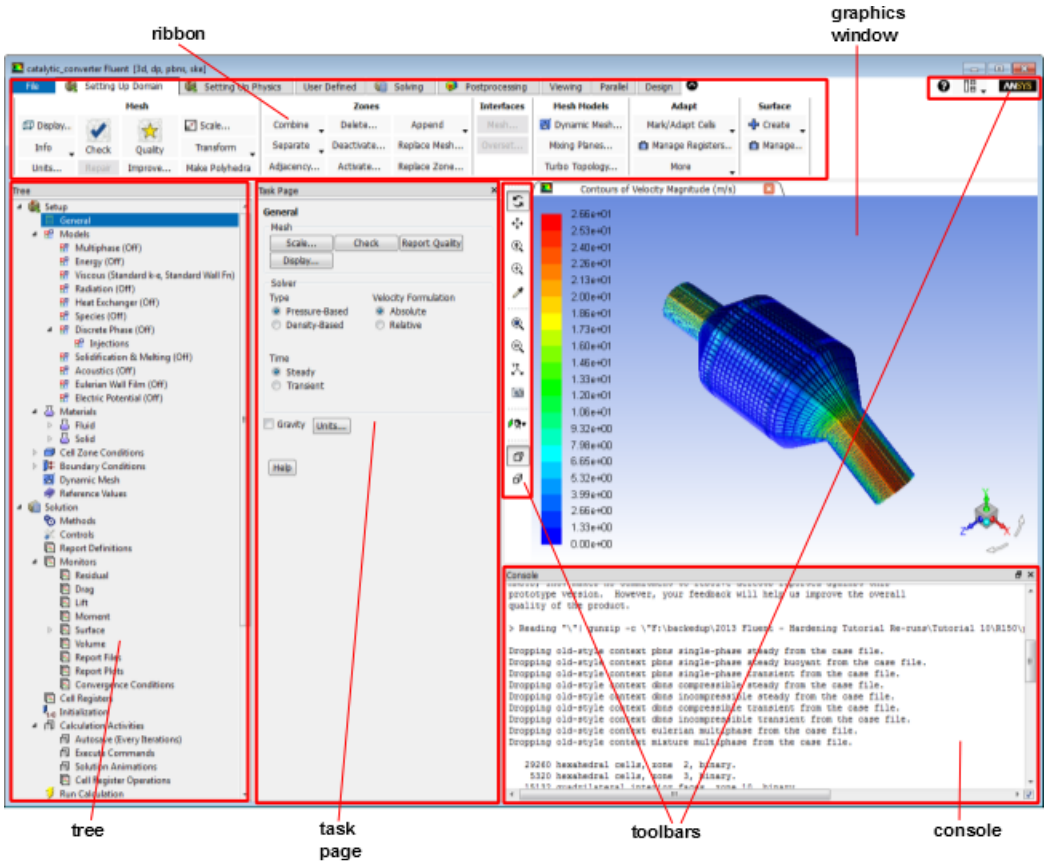
1.1. GUI Components

The graphical user interface (GUI) is made up of seven main components, which are described in detail in the subsequent sections: the ribbon, the tree, toolbars, task pages, a console, dialog boxes, and graphics windows. When you use the GUI, you will be interacting with one of these components at all times. The GUI will change depending on whether you are in meshing mode (as described in the Fluent Meshing User's Guide) or solution mode (as described in this guide, and as shown in Figure 1.1: The GUI Components (p. 2)). For details on how to switch between these modes, see Switching Between Meshing and Solution Modes in the Getting Started Guide.

Many of the GUI elements can be moved or tabbed together to suit your preferences. You can also modify attributes of the GUI (including colors and text fonts) to better match your platform environment. These are described in Customizing the Graphical User Interface (p. 22).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.1: The GUI Components



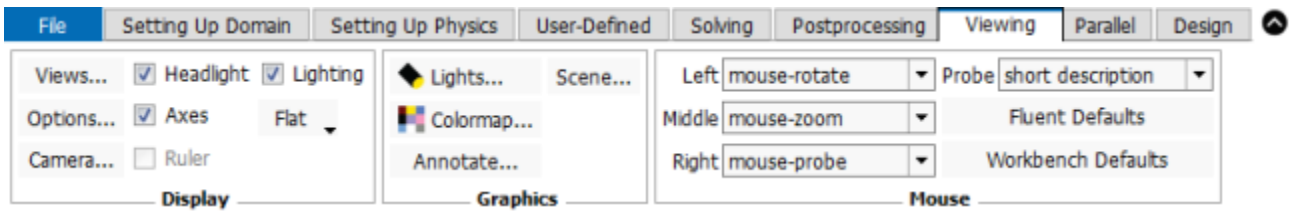
For additional information, see the following sections:

1.1.1. The Ribbon
1.1.2. The Tree
1.1.3. Graphics Windows
1.1.4. Toolbars
1.1.5. Task Pages
1.1.6. The Console
1.1.7. Dialog Boxes

1.1.1. The Ribbon

The ribbon, located at the top of the Fluent GUI, is the primary method for setting up and running your simulation. It facilitates access to the most commonly used items, with tabs nominally arranged in a left to right workflow for a typical simulation. Contents within each tab are grouped with related content and are organized logically to accommodate varied projects and priorities. You have the option to minimize the ribbon by clicking , located to the right of the ribbon tabs.

Figure 1.2: The Fluent Ribbon



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

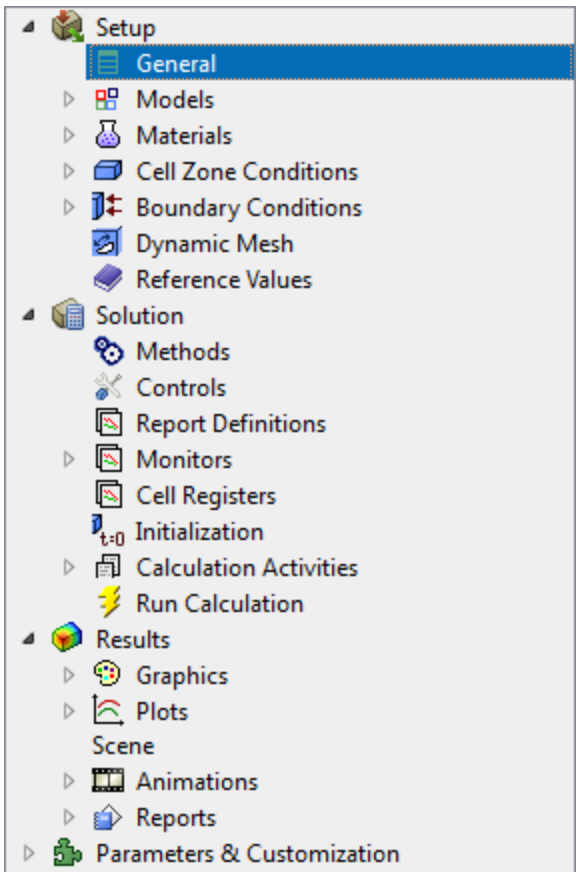
Often, making a selection in the ribbon opens a dialog box. In some instances, a ribbon selection opens a task page.

1.1.2. The Tree

The tree, located on the left side of the Fluent GUI, is an interactive representation of your project. Its items are listed in an order that complements the typical problem setup you follow in the ribbon. Branches can be expanded and collapsed to suit your preferences.

You can select multiple related items by left-clicking and dragging or using **Shift**/**Ctrl** + left-click. Right-clicking selected items in the tree provides a list of common commands. Typographical Conventions (p. lxix) shows the convention used to describe this right-click action, when it is referred to in the documentation.

Figure 1.3: The Fluent Tree



Double-clicking many of the branches under **Setup**, **Solution**, or **Results** displays the corresponding **Task Page** to the right of the tree. For additional information, see (Task Pages (p. 10)).

1.1.3. Graphics Windows

Graphics windows display the program's graphical output, and may be viewed within the Fluent application window or in separate windows. You can right-click the top of the graphics window to select whether you want new graphics windows to be tabbed or as additional sub-windows within the graphics window group space. By default, graphics windows viewed within the application window will be placed below the toolbar on the right, as shown in Figure 1.1: The GUI Components (p. 2).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using context menus in the graphics window you can access and modify settings directly on the model. Figure 1.4: Graphics Window Context Menu: Single-Selection (p. 4) and Figure 1.5: Graphics Window Context Menu: Multiple-Selection (p. 5) demonstrate some of the options available via right-click. *You can make multiple selections on the displayed graphics object by holding* ***Ctrl***.

Figure 1.4: Graphics Window Context Menu: Single-Selection

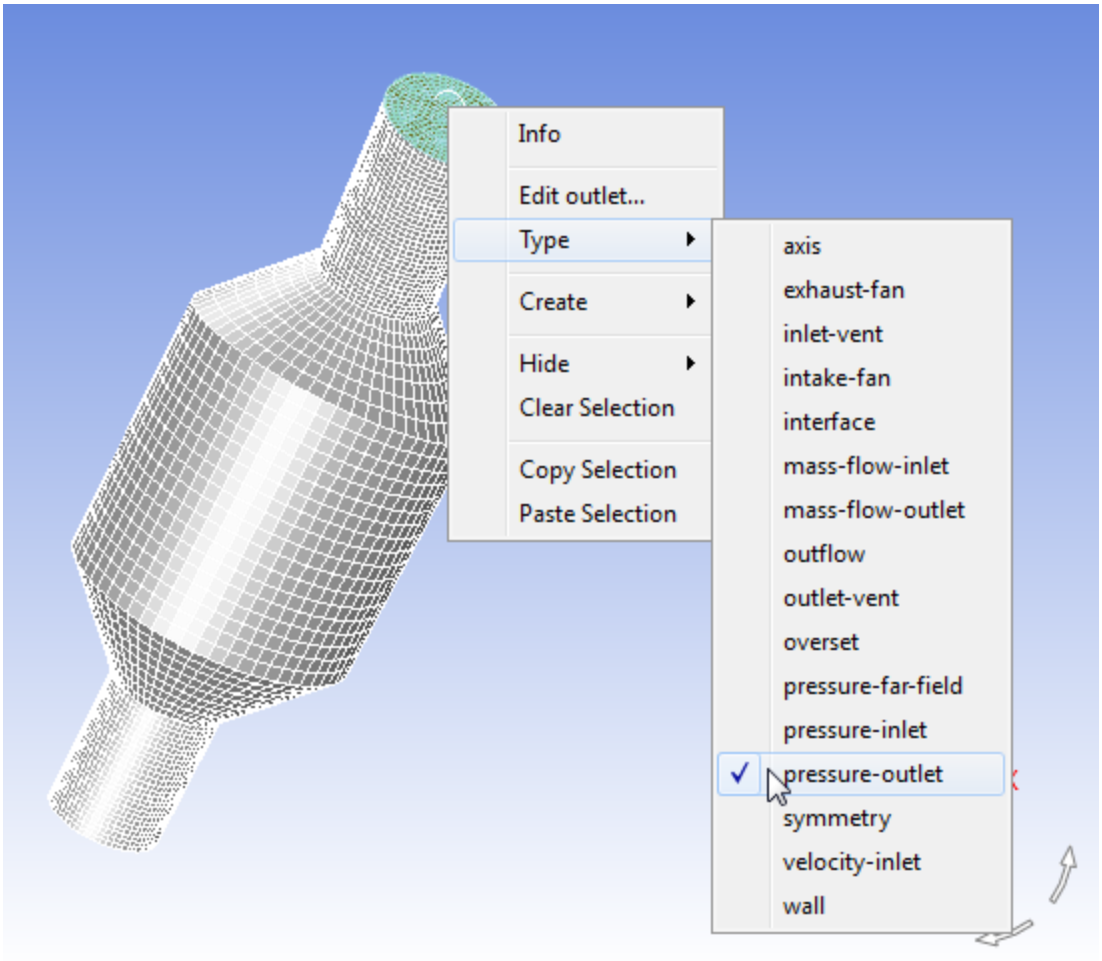


Figure 1.4: Graphics Window Context Menu: Single-Selection (p. 4) shows right-clicking the outlet (selection highlighted in green) and specifying it as a ***pressure-outlet***.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.5: Graphics Window Context Menu: Multiple-Selection

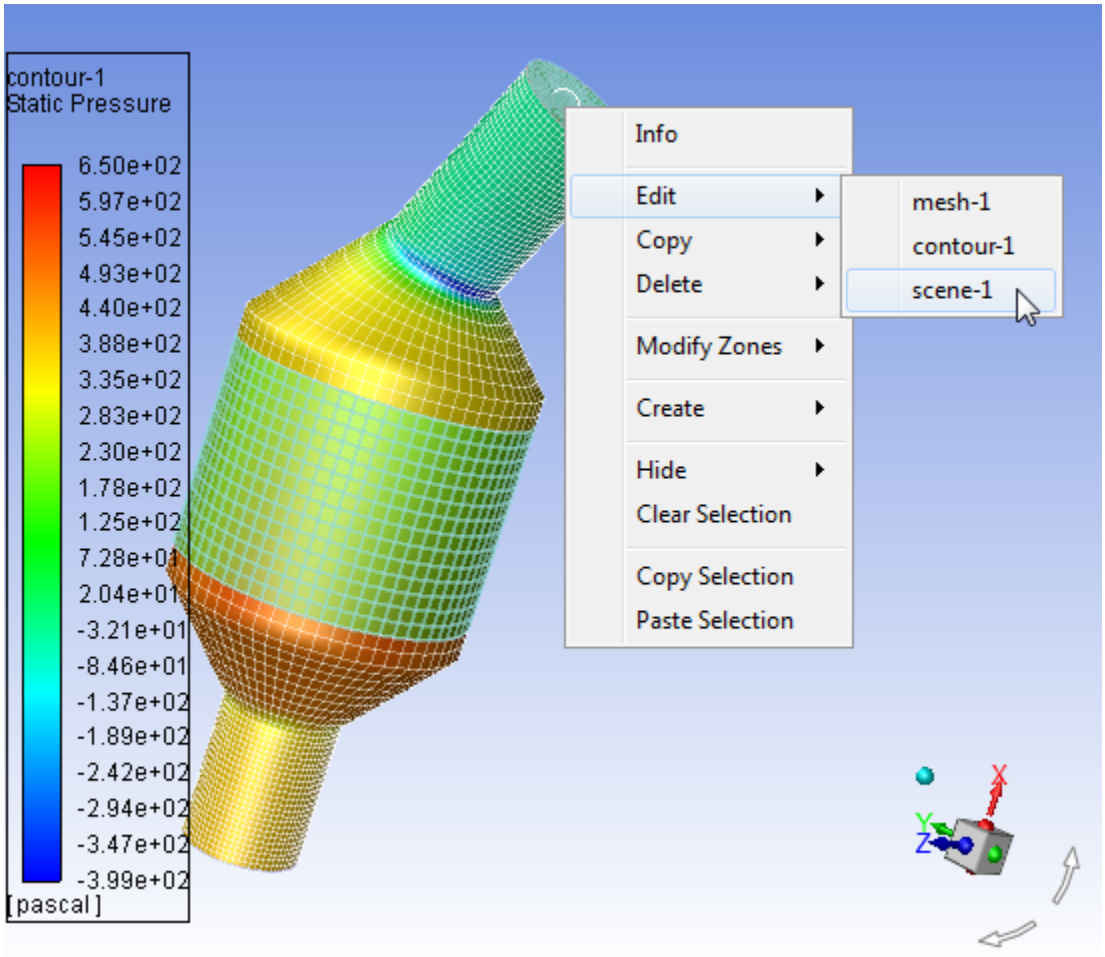


Figure 1.5: Graphics Window Context Menu: Multiple-Selection (p. 5) shows right-clicking the object when multiple surfaces are selected and a scene is displayed.

Note

There are a few situations where a right-click in the graphics window will not open a context menu:

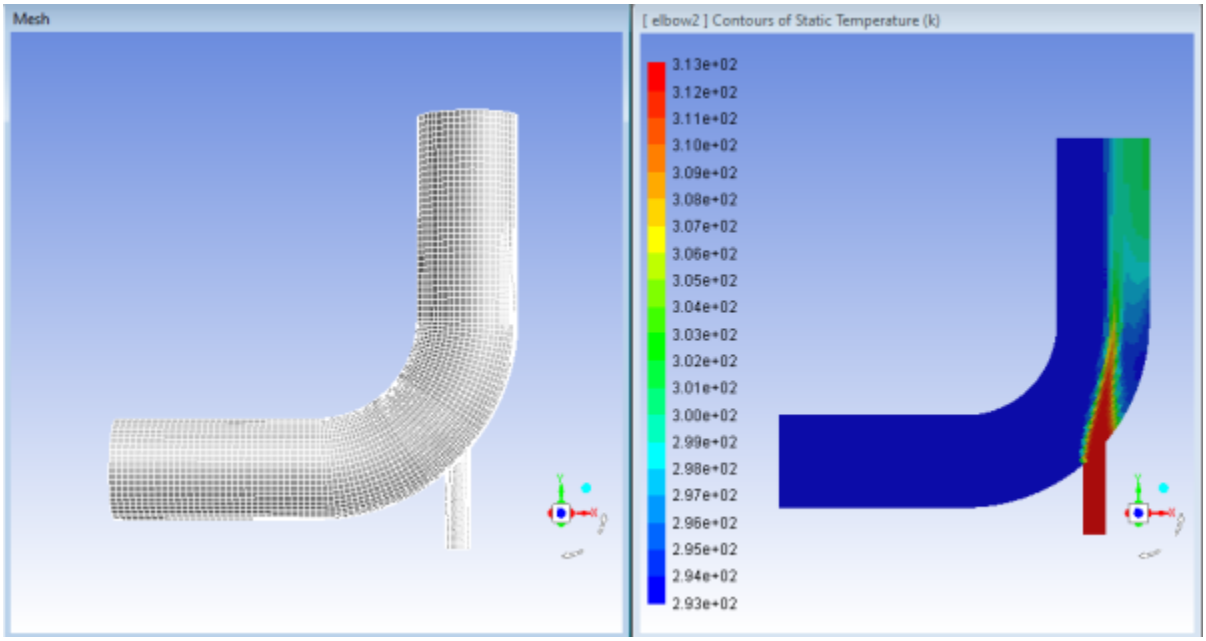
- Right-mouse-button (RMB) is set as **mouse-probe** and you right-click on the model when nothing is selected.
- RMB is set as **mouse-probe** and **long description** and you right-click anywhere in the graphics window when nothing is selected.
- RMB is set as **mouse-zoom**.

For additional information on mouse button settings, see Controlling the Mouse Button Functions (p. 1800).

In Figure 1.6: Displaying Two Graphics Windows (p. 6), two graphics windows are displayed by right-clicking the top of the graphics window and selecting **SubWindow View**, then creating two graphical displays and fitting them in the space accordingly.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.6: Displaying Two Graphics Windows



Important

You can change the text that appears in the graphics window title section by simply clicking in the area and editing it as you would in a text editor.

The **Display Options** dialog box (see Display Options Dialog Box (p. 2573)) can be used to change the attributes of the graphics window or to open another graphics window. In the **Viewing** tab (**Mouse** group box), you can set the action taken in the graphics window when a mouse button is pressed (see Modifying the Mouse Button Functions (p. 1801) for details).

Important

To cancel a display operation, press **Ctrl**+**c** while the data is being processed in preparation for graphical display. You cannot cancel the operation after the program begins to draw in the graphics window.

Important

It is possible that the graphics window may become **[Out of Date]** (which would be indicated at the top of the graphics window), if you make changes to items that are already displayed. To resolve this out of date state, right-click in the graphics window and click **Refresh Display** in the context menu that appears.

*If a context menu does not appear on a right-click of the graphics window, ensure that the right-mouse button is set to an action other than **mouse-zoom** or **mouse-probe** and **long description** (**Viewing** ribbon tab, **Mouse** group box).*

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Copy to Clipboard

Right-clicking the graphics window tab allows you to select **Copy to Clipboard**. This places a copy of the current picture displayed in the selected graphics window tab onto your clipboard. The size of your graphics window affects the size of the text fonts used in the picture.

1.1.4. Toolbars

The Fluent GUI includes toolbars located within the application window. These toolbars provide shortcuts to performing common tasks in Fluent. The toolbars include a standard toolbar and an objects toolbar.

1.1.4.1. The Standard Toolbar

The standard toolbar (Figure 1.7: The Standard Toolbar (p. 7)) contains options for getting help, arranging the graphical user interface, and visiting the ANSYS website.

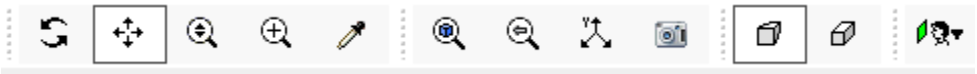
Figure 1.7: The Standard Toolbar

The following is a brief description of each of the standard toolbar options:

- **Help** allows you to access the Fluent User's Guide for help topics. For more information, see Using the GUI Help System (p. 23).
- **Arrange the workspace** provides you with several application window layout options. For example, you can choose to hide certain windows or have the graphics window be much larger than the other elements. These options also allow you to return to a standard layout if you accidentally rearrange (by dragging and dropping, or closing elements) the workspace in an undesirable manner.
- **Link to the ANSYS Website** opens a link to the ANSYS home page in your default browser.

1.1.4.2. The Graphics Toolbar

Figure 1.8: The Graphics Toolbar



The following tools are available in the graphics toolbar:

1.1.4.2.1. Pointer Tools
1.1.4.2.2. View Tools
1.1.4.2.3. Projection Tools
1.1.4.2.4. Display Options

1.1.4.2.1. Pointer Tools

The pointer tools (Figure 1.9: The Pointer Tools (p. 8)) allow you to modify the way in which you view your model or select objects in the graphics window.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.9: The Pointer Tools

The following is a description of each of the pointer tools:

- **Rotate View** lets you rotate your model about a central point in the graphics window. For more information, see Button Functions (p. 1800).
- **Pan** allows you to pan horizontally or vertically across the view using the left mouse button. For more information, see Button Functions (p. 1800).
- **Zoom In/Out** allows you to zoom in to and out of the model by holding the left mouse button down and moving the mouse down or up. For more information, see Button Functions (p. 1800). You can also roll the view by holding the left mouse button down and moving the mouse left or right.
- **Zoom to Area** allows you to focus on any part of your model. After selecting this option, position the mouse pointer at a corner of the area to be magnified, hold down the left mouse button and drag open a box to the desired size, and then release the mouse button. The enclosed area will then fill the graphics window. Note that you must drag the mouse to the right in order to zoom in. To zoom out, you must drag the mouse to the left. For more information, see Button Functions (p. 1800).
- **Print information about selected item** allows you to select items from the graphics windows and request information about displayed scenes. This behaves as a mouse probe button. For more information, see Button Functions (p. 1800).

1.1.4.2.2. View Tools

The view tools (Figure 1.10: The View Tools (p. 8)) allow you to modify the way your model is displayed in the graphics window.

Figure 1.10: The View Tools

The following is a description of each of the view tools:

- **Fit to Window** adjusts the overall size of your model to take maximum advantage of the graphics window's width and height.
- **Last View** allows you to revert to the displayed objects previous location and orientation in the graphics window.
- **Isometric view** contains a drop-down of views, allowing you to display the model from the direction of the vector equidistant to all three axes, as well as in different axes orientations.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Save Picture** allows you to capture an image of the active graphics window. For more information, see Saving Picture Files (p. 102).

1.1.4.2.3. Projection Tools

The projection tools (Figure 1.11: The Projection Tools (p. 9)) allow you to modify the perspective of objects in the graphics window.

Figure 1.11: The Projection Tools

The following is a brief description of each of the projection tools:

- **Perspective View** shows the 3D object on your 2D screen approximately how your eye would see it in real life. This button is only available when running Fluent in 3D.

- **Orthographic View** shows 3D objects to scale, ignoring perspective view, so that the displayed ruler accurately shows object size. This button is only available when running Fluent in 3D.

1.1.4.2.4. Display Options

The display options (Figure 1.12: The Display Options (p. 9)) contains predefined settings for how a mesh and objects are displayed.

Figure 1.12: The Display Options

The following is a brief description of the display options:

- **Mesh Display Configuration**

  contains options for changing how the mesh displays in the graphics window.

  – **Meshing**

    draws mesh on edges and faces of the outline surfaces, colored by their zone ID with lighting enabled.

  – **Solution**

    draws mesh on edges and faces of the outline surfaces, colored by their zone type with lighting enabled.

  – **Post Processing**

    draws the object outline with lighting disabled.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **Classic**

  draws mesh on all edges of the outline surfaces.

1.1.5. Task Pages

Task pages appear on the right side of the tree when certain branches are double-clicked. They provide access to more advanced settings than are often available through a right-click of the same branch or sub-branches. The expected workflow is that you travel down the tree, using a right-click most branches and sub-branches to access the settings and controls required for to solve your problem.

Some of your setup will occur in dialog boxes, while others in task pages. For example, if you double-click **General** in the tree, the General Task Page (p. 2160) is displayed. Global settings are made in this task page, which are saved to the case definition.

Each task page has a **Help** button. Clicking this button opens the related help topic in the Task Page Reference Guide (p. 2159). See Using the GUI Help System (p. 23) for more information.

1.1.6. The Console

The console is located below the graphics window, as shown in Figure 1.1: The GUI Components (p. 2). Fluent communicates with you through the console. It is used to display various kinds of information (that is, messages relating to meshing or solution procedures, and so on). Console text is colored based on whether it is user-input, standard output, or a warning message. Fluent saves a certain amount of information that is written to the console into memory. You can review this information at any time by using the scroll bar on the right side of the console. You can enable/disable autoscrolling using the check box at the bottom right of the console to either have the cursor jump to the latest printed content or stay where it is. The size of the console can be adjusted by raising or lowering the bottom frame of the graphics window.

The console is similar in behavior to "xterm" or other Linux command shell tools, or to the MS-DOS Command Prompt window on Windows systems. It allows you to interact with the TUI menu. More information on the TUI can be found in Text User Interface (TUI) (p. 27).

The console accepts a "break" command (pressing **Ctrl**+**c** at the same time) to let you interrupt the program while it is working. It also lets you perform text copy and paste operations between the console and other X Window (or Windows) applications that support copy and paste. The following steps show you how to perform a copy and paste operation on a Windows system:

1. Move the pointer to the beginning of the text to be copied.
2. Press and hold down the left mouse button.
3. Move the pointer to the end of the text (text should be highlighted).
4. Release the left mouse button.
5. Press the **Ctrl** and `<Insert>` keys at the same time.
6. Move the pointer to the target window and click the left mouse button.
7. Press the **Ctrl**+**v** keys at the same time.

On a Linux system, you will follow the steps below to copy text to the clipboard:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Move the pointer to the beginning of the text to be copied.
2. Press and hold down the left mouse button.
3. Move the pointer to the end of the text (text should be highlighted).
4. Release the left mouse button.
5. Move the pointer to the target window.
6. Press the middle mouse button to "paste" the text.

1.1.7. Dialog Boxes

There are two types of dialog boxes in Fluent. Some dialog boxes are used to perform simple input/output tasks, such as issuing warning and error messages, or asking a question requiring a yes or no answer. Other forms of dialog boxes allow you to perform more complicated input tasks.

A dialog box is a separate "temporary" window that appears when Fluent needs to communicate with you, or when various types of input controls are employed to set up your case. The types of controls you will see are described further in this section.

When you have finished entering data in a dialog box's controls, you must apply the changes you have made, or cancel the changes, if desired. For this task, each dialog box falls into one of two behavioral categories, depending on how it was designed.

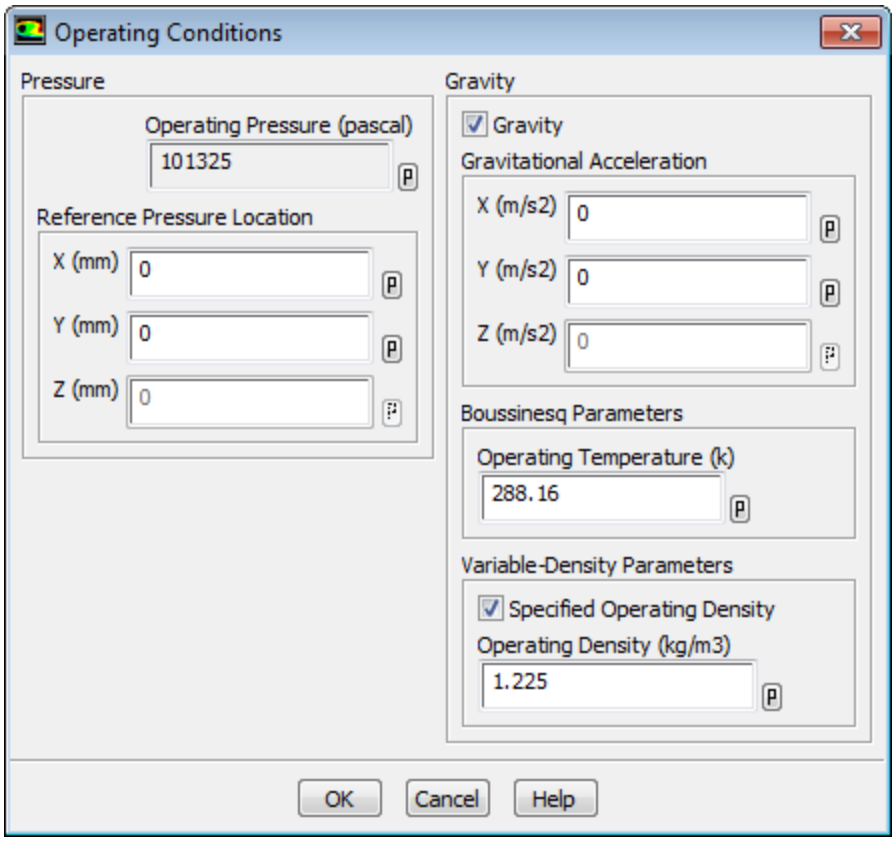
The first category of dialog boxes is used in situations where it is desirable to apply the changes and immediately close the dialog box. This type of dialog box includes an **OK** and a **Cancel** button that function as described below:

OK
applies any changes you have made to the dialog box, then closes the dialog box.

Cancel
closes the dialog box, ignoring any changes you have made.

An example of this type of dialog box is shown in the following figure:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



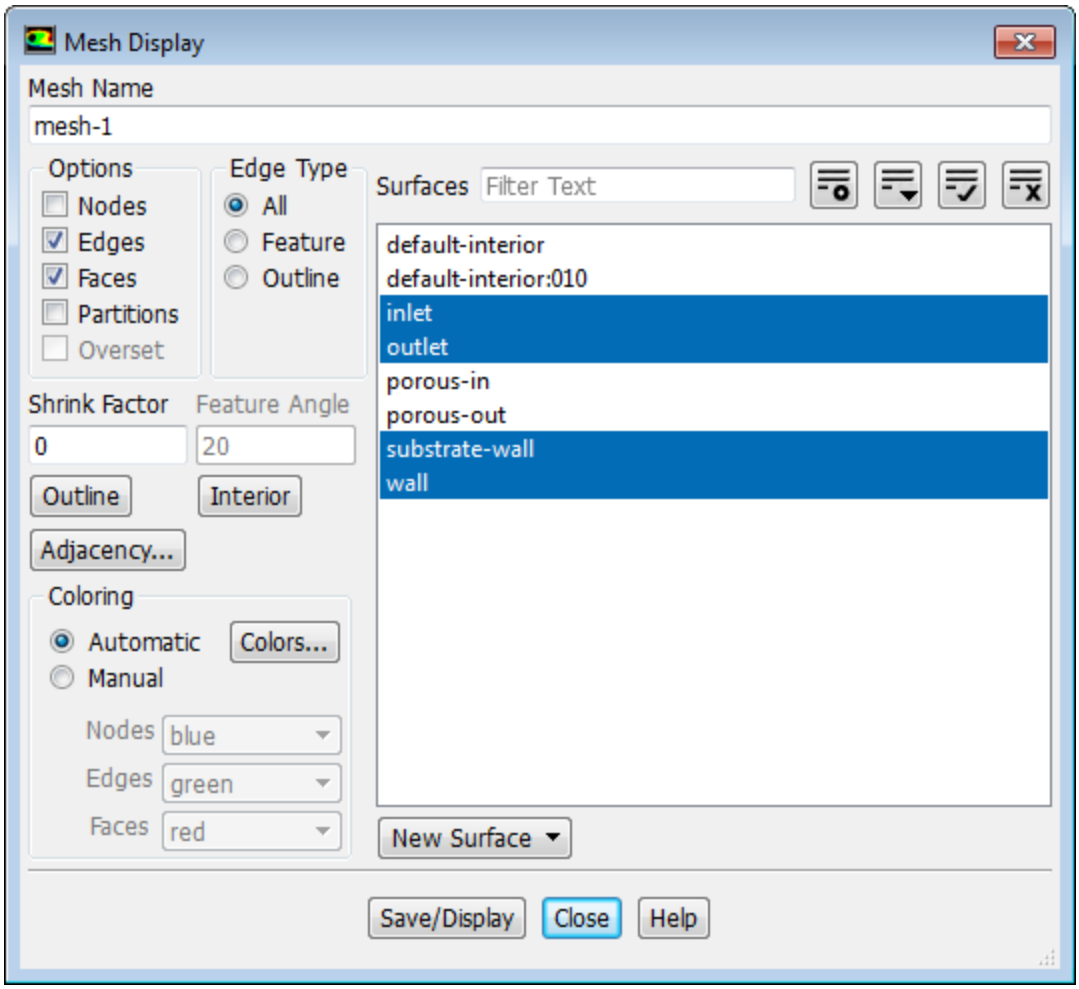
The other category of dialog boxes is used in situations where it is desirable to keep the dialog box displayed on the screen after changes have been applied. This makes it easy to quickly go back to that dialog box and make more changes. Dialog boxes used for postprocessing and mesh adaption often fall into this category. This type of dialog box typically includes an **Apply** button and a **Close** button as described below:

Apply
applies any changes you have made to the dialog box, but does not close the dialog box. The name of this button is often changed to something more descriptive. For example, many of the postprocessing dialog boxes use the name **Display** for this button, and the adaption dialog boxes use the name **Adapt**.

Close
closes the dialog box.

An example of this type of dialog box is shown in the following figure:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



All dialog boxes include the following button used to access ANSYS Help:

Help
displays information about the controls in the dialog box. The help information will appear in the ANSYS Help Viewer.

Many dialog boxes also present additional buttons to accomplish specific tasks or open additional dialog boxes. You can find descriptions of the functions of these additional buttons in Ribbon Reference Guide (p. 2633) or Task Page Reference Guide (p. 2159).

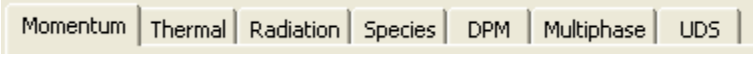
Note

Dialog boxes containing lists are expandable.

1.1.7.1. Input Controls

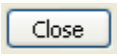
Each type of input control utilized by the dialog boxes is described below. Note that the examples shown here are for a Windows system; if you are working on a Linux system, your dialog box controls may look slightly different, but they will work exactly as described here.

1.1.7.1.1. Tabs



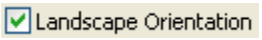
Much like the tabs on a notebook divider, tabs in dialog boxes are used to mark the different sections into which a dialog box is divided. A dialog box that contains many controls may be divided into different sections to reduce the amount of screen space it occupies. You can access each section of the dialog box by "clicking" the left mouse button on the corresponding tab. A click is one press and release of the mouse button.

1.1.7.1.2. Buttons



A button, also referred to as a push button, is used to perform a function indicated by the button label. To activate a button, place the pointer over the button and click the left mouse button.

1.1.7.1.3. Check Boxes

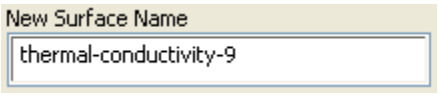


A check box, also referred to as a check button, is used to enable / disable an item or action indicated by the check box label. Click the left mouse button on the check box to toggle the state.

1.1.7.1.4. Radio Buttons

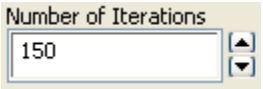
Radio buttons are a set of check boxes with the condition that only one can be set in the "on" position at a time. When you click the left mouse button on a radio button, it will be turned on, and all others will be turned off. Radio buttons appear either as diamonds (in Linux systems) or as circles (as shown above).

1.1.7.1.5. Text Entry Boxes



A text entry box lets you type text input. It will often have a label associated with it to indicate the purpose of the entry.

1.1.7.1.6. Integer Number Entry Boxes



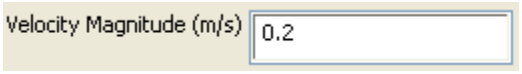
An integer number entry box is similar to a text entry except it only allows integer numbers to be entered (for example, 10, -10, 50000 and 5E4). You may find it easier to enter large integer numbers using scientific notation. For example, you could enter `350000` or `3.5E5`.

The integer number entry also has arrow buttons that allow you to easily increase or decrease its value. For most integer number entry controls, the value will be increased (or decreased) by one when you click an arrow button. You can increase the size of the increment by holding down a keyboard key while clicking the arrow button. The keys used are shown below:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Key	Factor of Increase
Shift	`10`
Ctrl	`100`

1.1.7.1.7. Real Number Entry Boxes

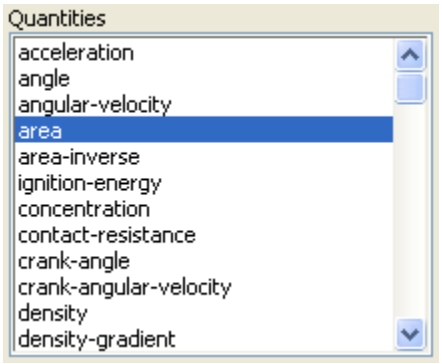


A real number entry box is similar to a text entry, except it only allows real numbers to be entered (for example, 10, -10.538, 50000.45 and 5.72E-4). In most cases, the label will show the units associated with the real number entry.

1.1.7.1.8. Filter Text Entry Boxes

The filter text entry box allows you to search and organize a list using a text string. The text string can include wildcards, which allow you to perform pattern matching. For example, searching `*let` finds surfaces such as in*let*s and out*let*s. If you have walls separated by a number, you can search `example-wall-?-23`, which would show `example-wall-1-23`, `example-wall-2-23`, and so on.

1.1.7.1.9. Single-Selection Lists

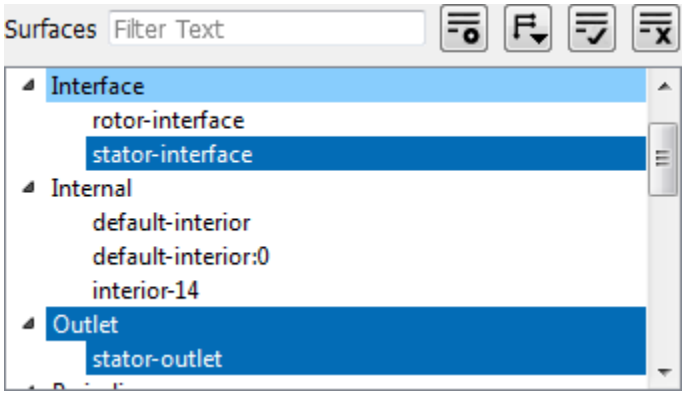


A single-selection list presents a list of items, with each item printed on a separate line. You can select an item by placing the pointer over the item line and clicking with the left mouse button. The selected item will become highlighted. Selecting another item will deselect the previously selected item in the list.

Many dialog boxes will also accept a double-click in order to invoke the dialog box action that is associated with the list selection (see information on the dialog box of interest for more details).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.1.7.1.10. Multiple-Selection Lists



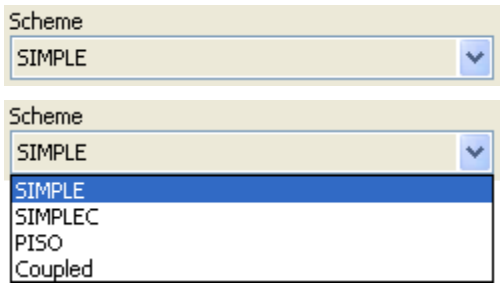
A multiple-selection list is similar to a single-selection list, except it allows for more than one selected item at a time. When you click the left mouse button on an item, its selection state will toggle. Clicking an unselected item will select it. Clicking a selected item will deselect it.

To select a range of items in a multiple-selection list, you can select the first desired item, and then select the last desired item while holding down the **Shift** key. The first and last items, and all the items between them, will be selected. You can also click and drag the left mouse button to select multiple items.

There are several small buttons in the upper right corner of the multiple selection list to accelerate selecting or deselecting items from the selection list.

- Click to display all of the items that are currently selected.
- Click or to toggle to a tree view of the selection list, and thereby group items by categories such as name or surface type. You can select or deselect all the items in a particular group by clicking the top-level branch. For example, if you have the surfaces grouped by **Surface Type**, clicking **Inlet** will select or deselect all inlets. Note that once you switch to a tree view, the icon changes to or , respectively.
- Click to select all the items displayed in the selection list.
- Click to deselect all the items displayed in the selection list.

1.1.7.1.11. Drop-Down Lists



A drop-down list is a hidden single-selection list that shows only the current selection to save space.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you want to change the selection, follow the steps below:

1. Click the arrow button to display the list.

2. Place the pointer over the new list item.

3. Click the left mouse button on the item to make the selection and close the list.

If you want to stop the selection operation while the list is displayed, you can move the pointer anywhere outside the list and click the left mouse button.

1.1.7.1.12. Scales

A scale is used to select a value from a predefined range by moving a slider. The number shows the current value. You can change the value by clicking the arrow buttons, or by following one of the procedures below:

1. Place the pointer over the slider.

2. Press and hold down the left mouse button.

3. Move the pointer along the slider bar to change the value.

4. Release the left mouse button.

or

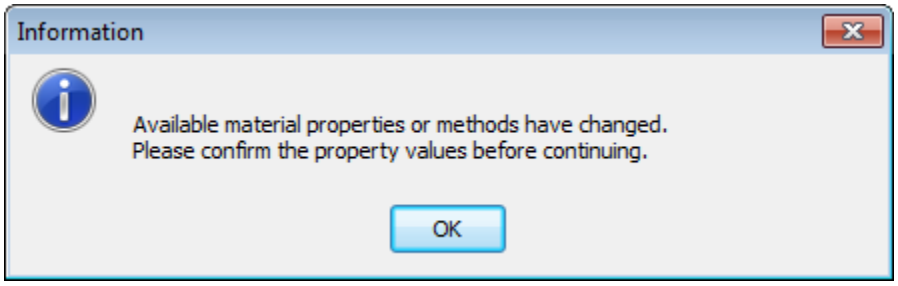
1. Place the pointer over the slider and click the left mouse button.

2. Using the arrow keys on the keyboard, move the slider bar left or right to change the value.

1.1.7.2. Types of Dialog Boxes

The following sections describe the various types of dialog boxes.

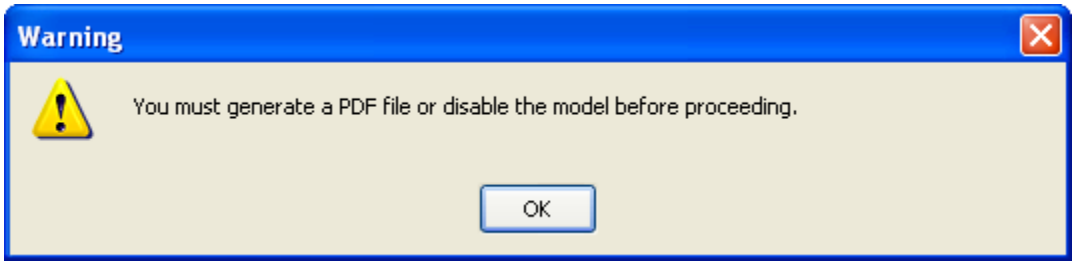
1.1.7.2.1. Information Dialog Boxes



The **Information** dialog box is used to report some information that Fluent thinks you should know. After you have read the information, you can click the **OK** button to close the dialog box.

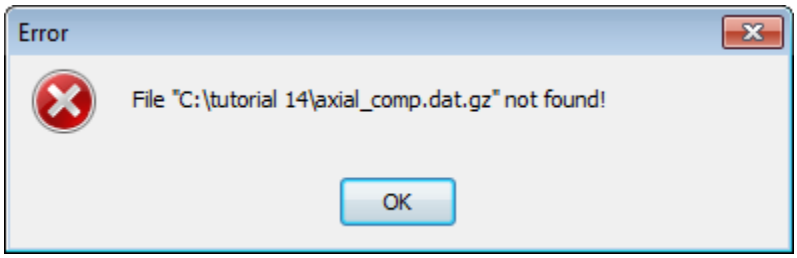
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.1.7.2.2. Warning Dialog Boxes



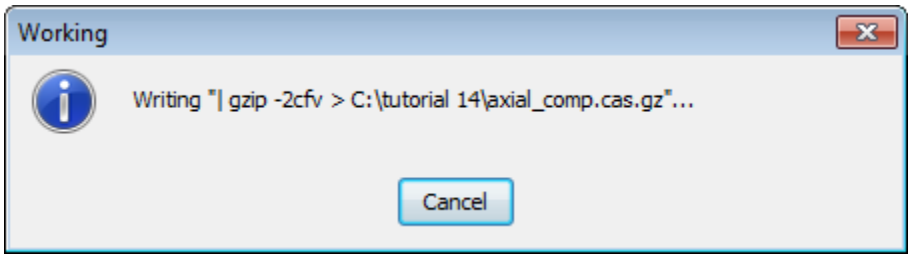
The **Warning** dialog box is used to warn you of a potential problem or deliver an important message. Your control of Fluent will be suspended until you acknowledge the warning by clicking the **OK** button.

1.1.7.2.3. Error Dialog Boxes



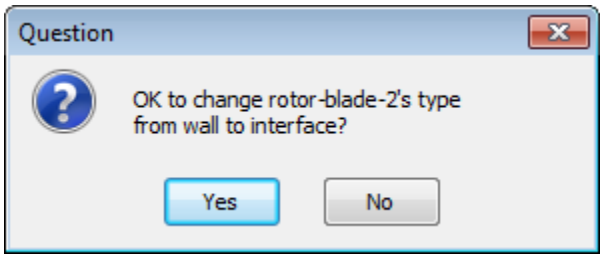
The **Error** dialog box is used to alert you of an error that has occurred. After you have read the error information, you can click the **OK** button to close the dialog box.

1.1.7.2.4. The Working Dialog Box



The **Working** dialog box is displayed when Fluent is busy performing a task. This is a special dialog box, because it requires no action by you. It is there to let you know that you must wait. When the program is finished, it will close the dialog box automatically. You can, however, abort the task that is being performed by clicking the **Cancel** button.

1.1.7.2.5. Question Dialog Box



The **Question** dialog box is used to ask you a question. Sometimes the question will require a **Yes** or **No** answer, while other times it will require that you either allow an action to proceed (**OK**) or **Cancel** the action. You can click the appropriate button to answer the question.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

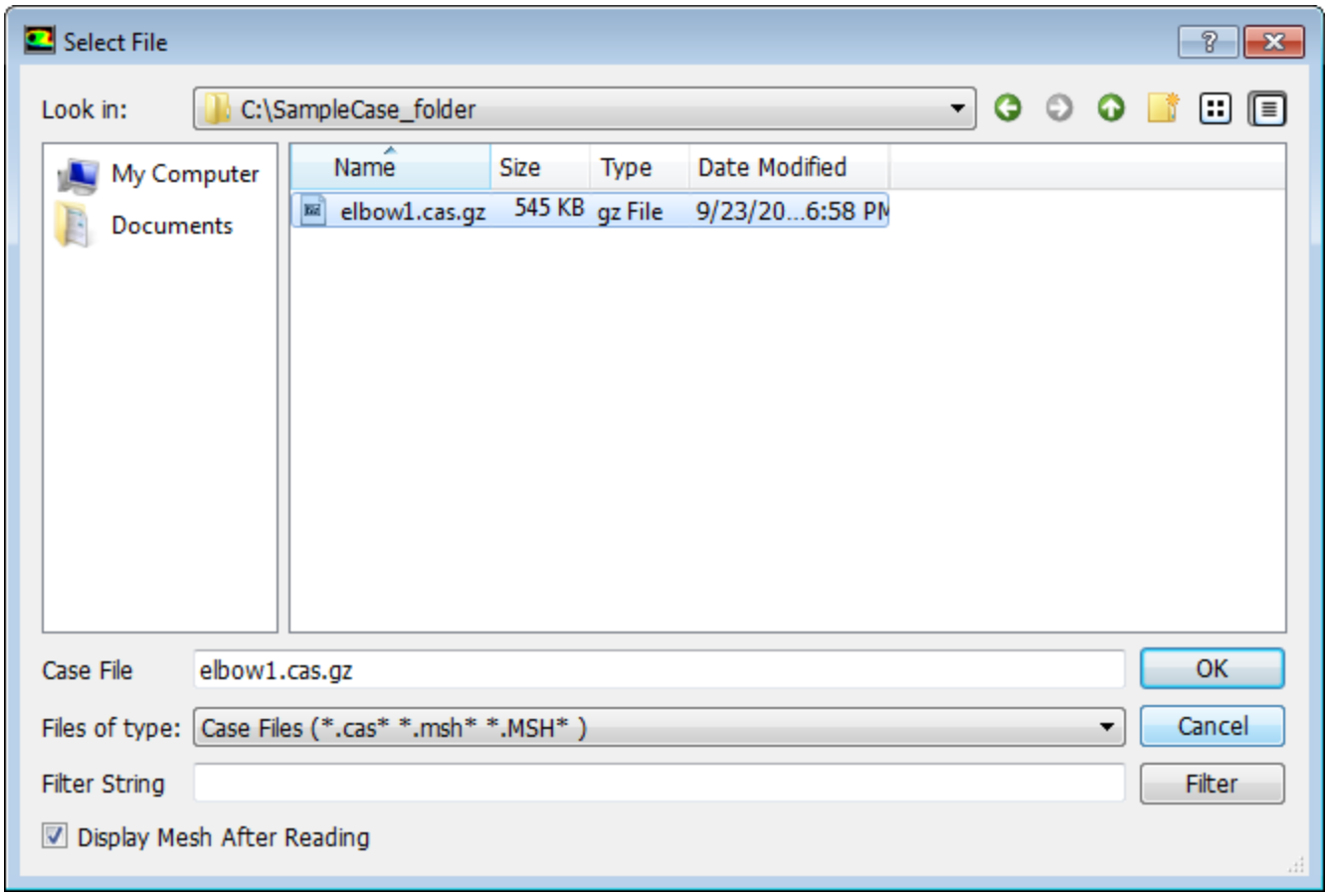
1.1.7.2.6. The Select File Dialog Box

File selection is accomplished using the **Select File** dialog box (The Select File Dialog Box (Windows) (p. 19) or The Select File Dialog Box (Linux) (p. 19)).

1.1.7.2.6.1. The Select File Dialog Box (Windows)

File selection on Windows systems is accomplished using the standard Windows **Select File** dialog box (Figure 1.13: The Select File Dialog Box for Windows (p. 19)).

Figure 1.13: The Select File Dialog Box for Windows



See documentation regarding your Windows system for further instructions on file selection.

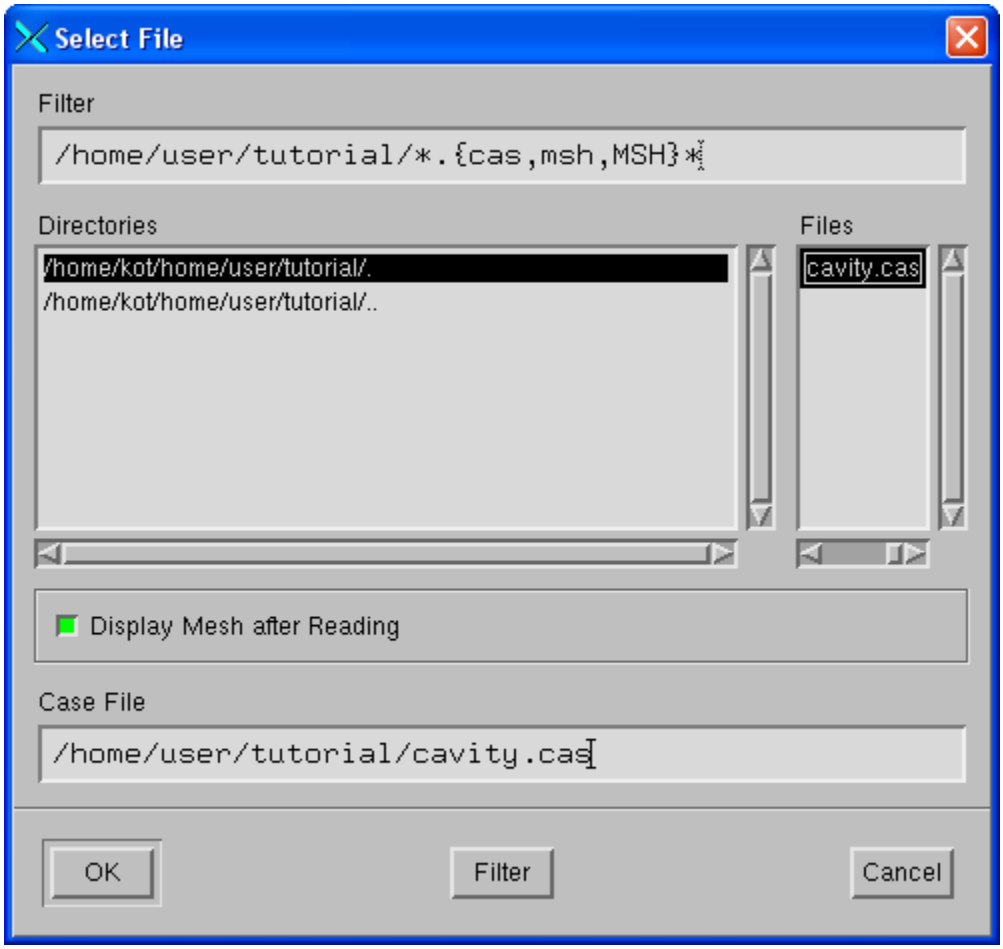
1.1.7.2.6.2. The Select File Dialog Box (Linux)

For Linux systems, note that the appearance of the **Select File** dialog box will not always be the same.

The version shown in Figure 1.14: The Select File Dialog Box for Linux Platforms (p. 20) will appear in almost all cases, but it will be different if you are loading external data files for use in an XY plot (see Including External Data in the Solution XY Plot (p. 1844) for more information). In such cases, the dialog box will look like Figure 1.15: Another Version of the Select File Dialog Box for Linux Platforms (p. 21).

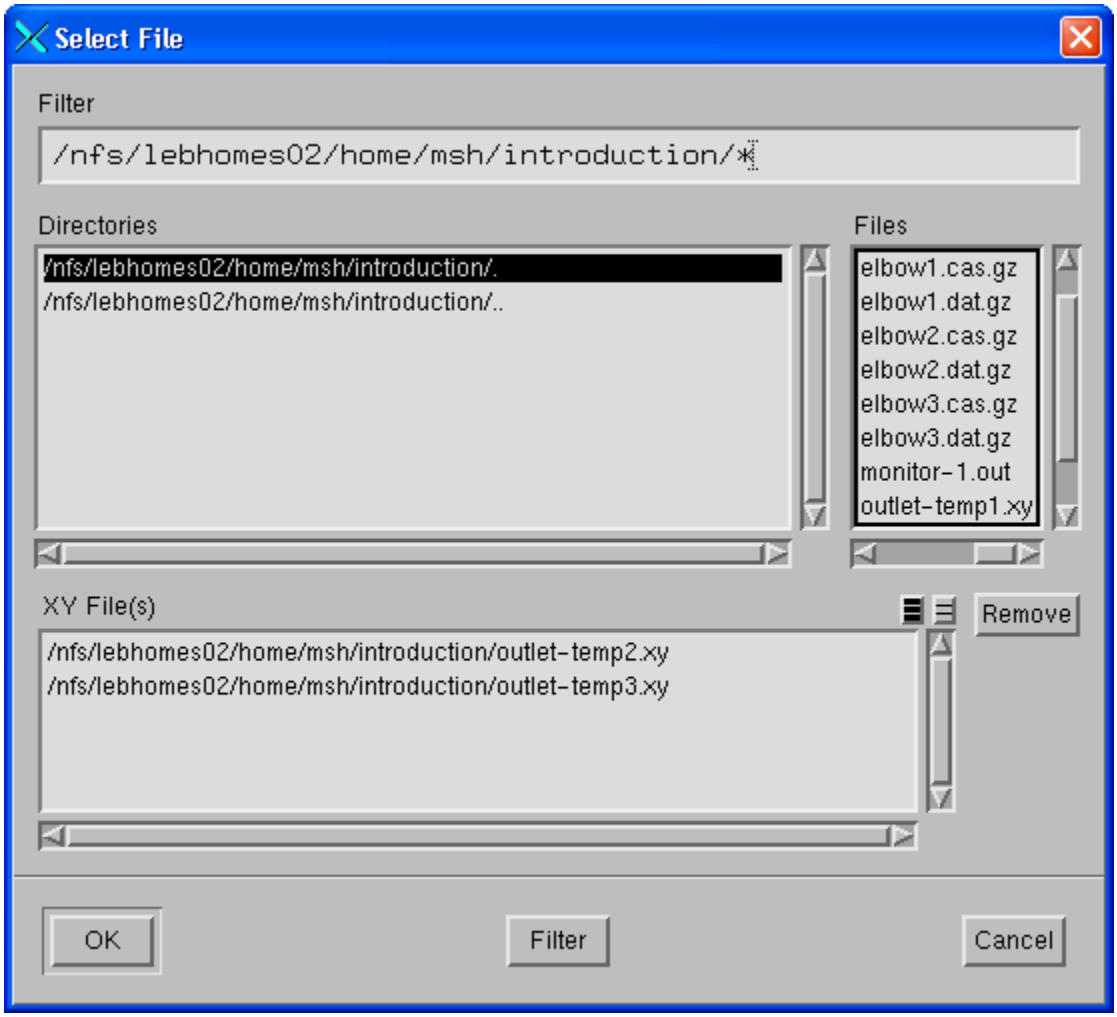
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.14: The Select File Dialog Box for Linux Platforms



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.15: Another Version of the Select File Dialog Box for Linux Platforms



The steps for file selection are as follows:

1. Go to the appropriate directory. You can do this in two different ways:

   - Enter the path to the desired directory in the **Filter** text entry box and then press the **Enter** key or click the **Filter** button. Be sure to include the final / character in the pathname, before the optional search pattern (described below).

   - Double-click a directory, and then a subdirectory, and so on, in the **Directories** list until you reach the directory you want. You can also click once on a directory and then click the **Filter** button, instead of double-clicking. Note that the "**.**" item represents the current directory and the "**..**" item represents the parent directory.

2. Specify the file name by selecting it in the **Files** list or entering it in the **File** text entry box (if available) at the bottom of the dialog box. The name of this text entry box will change depending on the type of file you are selecting (**Case File**, **Journal File**, and so on).

Important

Note that if you are searching for an existing file with a nonstandard extension, you may need to modify the "search pattern" at the end of the path in the **Filter** text entry box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For example, if you are reading a data file, the default extension in the search path will be `*.dat*`, and only those files that have a `.dat` extension will appear in the **Files** list. If you want files with a `.DAT` extension to appear in the **Files** list, you can change the search pattern to `*.DAT*`. If you want all files in the directory to be listed in the **Files** list, enter just `*` as the search pattern.

3. If you are reading a mesh or case file, use the **Display Mesh after Reading** option to specify whether you want Fluent to automatically display the mesh after the file is read. All of the boundary zones will be displayed, except for the interior zones of 3D geometries. The default status of this option (that is, enabled or disabled) is determined by your decision regarding the **Display Mesh After Reading** option in Fluent Launcher.

4. If you are using the **Replace Mesh...** option in the **Setting Up Domain** tab (**Zones** group box) to replace a mesh with existing data, you can enable the **Interpolate Data Across Zones** option to interpolate the data across cell zones. This option is appropriate when the matching zone pairs (that is, the zones with the same names in both the current mesh and the replacement mesh) do not have the same interior zone boundaries. See Replacing the Mesh (p. 222) for details.

5. If you are reading multiple XY-plot data files, the selected file will be added to the list of **XY File(s)**. You can choose another file, following the instructions above, and it will also be added to this list. (If you accidentally select the wrong file, you can choose it in the **XY File(s)** list and click the **Remove** button to remove it from the list of files to be read.) Repeat until all of the desired files are in the **XY File(s)** list.

6. If you are writing a case, data, or radiation file, use the **Write Binary Files** check box to specify whether the file should be written as a text or binary file. You can read and edit a text file, but it will require more storage space than the same file in binary format. Binary files take up less space and can be read and written by Fluent more quickly.

7. Click the **OK** button to read or write the specified file. Shortcuts for this step are as follows:

   - If your file appears in the **Files** list *and* you are *not* reading an XY file, double-click it instead of just selecting it. This will automatically activate the **OK** button. (If you are reading an XY file, you will always have to click **OK** yourself. Clicking or double-clicking will just add the selected file to the **XY File(s)** list.)

   - If you entered the name of the file in the **File** text entry box, you can press the **Enter** key instead of clicking the **OK** button.

1.2. Customizing the Graphical User Interface

You may want to customize the graphical user interface by changing the way that the various elements are arranged. This can be achieved by "dragging" elements and "dropping" them at a new location. For example, certain items can be tabbed on top of other items, such as the graphics window on top of the console. The items that you can move include the console, the task page, and the toolbars.

To restore items that you intentionally or unintentionally closed, right-click the top toolbar to restore those items. You can also click to select one of the predefined layouts and restore missing items.

You may want to customize the graphical user interface by changing attributes such as text color, background color, and text fonts. The program will try to provide default text fonts that are satisfactory for your platform's display size, but in some cases customization may be necessary if the default text fonts make the GUI too small or too large on your display, or if the default colors are undesirable.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The GUI in Fluent is based on the **Qt Toolkit**. If you are unfamiliar with the **Qt Toolkit**, refer to any documentation you may have that describes how to use the **Qt Toolkit** or application. The graphical attributes can be modified in a Qt stylesheet file named `cxdisplay.qss` and placed in your home directory.

1.3. Using the GUI Help System

Fluent includes an integrated help system that provides easy access to the program documentation. Through the graphical user interface, you have the entire User's Guide and other documentation available to you with the click of a mouse button. The User's Guide and other manuals are displayed in the help viewer, which allows you to use hypertext links and the browser's search and index tools to find the information you need.

There are many ways to access the information contained in the online help. You can get reference information from within a task page or dialog box, or (on Linux machines) request context-sensitive help for a particular task page item or dialog box. You can also go to the User's Guide contents page or the viewer index, and use the hypertext links to find the information you are looking for. In addition to the User's Guide, you can also access the other Fluent documentation (for example, the Theory Guide or Fluent Customization Manual).

Note that the last two chapters of the User's Guide (Task Page Reference Guide (p. 2159) and Ribbon Reference Guide (p. 2633)) are also referred to as the Reference Guide, and contain a description of each task page, and dialog box.

The sections that follow provide information on how to get help for a task page or dialog box, and brief descriptions of the help menu items in Fluent. For more information, click the **Help** menu in the viewer. See also, Using Help.

1.3.1. Task Page and Dialog Box Help
1.3.2. Context-Sensitive Help (Linux Only)
1.3.3. Opening the User's Guide Table of Contents
1.3.4. Opening the Reference Guide
1.3.5. Help on Help
1.3.6. Accessing Printable (PDF) Manuals
1.3.7. Help for Text Interface Commands
1.3.8. Accessing the Customer Portal Web Site
1.3.9. Obtaining License Use Information
1.3.10. Version and Release Information

1.3.1. Task Page and Dialog Box Help

To get help about a task page or dialog box that you are currently using, click the **Help** button in the task page or dialog box. The help viewer will open to the section of the User's Guide that explains the function of each item in the task page or dialog box. In this section, you will also find hypertext links to more specific section(s) of the User's Guide that discuss how to use the task page or dialog box and provide related information.

1.3.2. Context-Sensitive Help (Linux Only)

If you want to find out how or when a particular task page item or dialog box is used, you can use the context-sensitive help feature. Select the **Context-Sensitive Help** item in the help menu (accessed by clicking).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

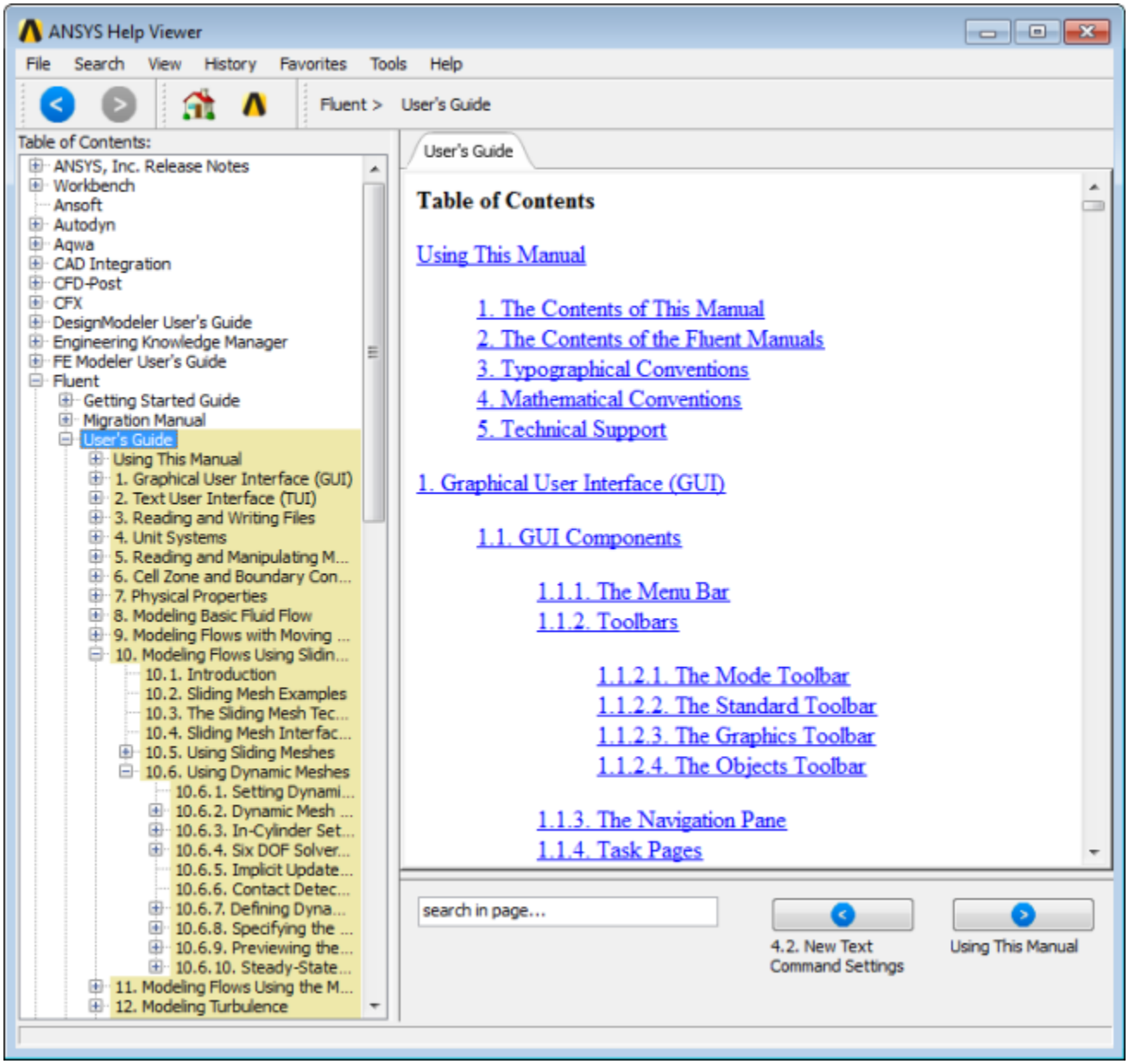
With the resulting question-mark cursor, select an item from a drop-down menu. The help viewer will open to the section of the User's Guide that discusses the selected item.

1.3.3. Opening the User's Guide Table of Contents

When in solution mode, you can view the User's Guide table of contents by selecting the **User's Guide Contents...** menu item in the help menu (accessed by clicking). Note that this menu item will open the Fluent Meshing User's Guide table of contents when you are in meshing mode.

Selecting this item will open the help viewer to the contents page of the User's Guide (Figure 1.16: The Fluent User's Guide Contents Page (p. 24)). Each item in the table of contents is a hypertext link that you can click to view that chapter or section.

Figure 1.16: The Fluent User's Guide Contents Page



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In addition, the Fluent documents are listed in the **Table of Contents** of the help viewer. These listings can be expanded by clicking the icon to the left of the document title ( ). The expanded list will contain hyperlinks to particular chapters or sections, and may also be expandable. For more information on navigation within the help viewer, refer to the information available from the **Help** menu in the viewer itself.

1.3.4. Opening the Reference Guide

To open the help system to the first page of the Reference Guide, which contains information about each dialog box or task page item, do the following in the help viewer:

1. Expand the Fluent **User's Guide** in the left pane of the viewer by clicking the icon to the left of the document title.
2. Scroll down to the **Task Page Reference Guide** or **Menu Reference Guide** item in the left pane of the viewer and click the one of interest.
3. Use the hyperlinks in the main viewer window to find the topic of interest, or, expand the item in the left pane of the viewer and scroll to the topic of interest.

1.3.5. Help on Help

Help( ) → **Using Help...**

When you select **Using Help...** from the help menu (accessed by clicking), the help viewer will open to the beginning of this section.

You can obtain information about using online help by selecting the **Using Help...** menu item in the **Help** drop-down menu in the help viewer itself. See also, Using Help.

1.3.6. Accessing Printable (PDF) Manuals

You can download the PDF documentation from the ANSYS Customer Portal (support.ansys.com/doc-downloads).

If you do not have Acrobat Reader, you can download it for free from Adobe (www.adobe.com).

See the Getting Started Guide for more information.

1.3.7. Help for Text Interface Commands

There are two ways to find information about text interface commands. You can either go to the Text Command List (which can be accessed in the **Contents** tab of the help viewer, as described in Opening the User's Guide Table of Contents (p. 24)), or use the text interface help system described in Using the Text Interface Help System (p. 36).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.3.8. Accessing the Customer Portal Web Site

You can access the ANSYS Customer Portal web site by selecting the **Online Technical Resources...** option in help menu (accessed by clicking). You can also access the ANSYS Customer Portal by clicking the button in ANSYS Help Viewer.

Help( ) → **Online Technical Resources...**

Fluent will direct your web browser to the appropriate web address.

You can download all ANSYS product documentation, including Fluent documentation, from the ANSYS Customer Portal web site. The documentation is available from the ANSYS Customer Portal in PDF format and is appropriate for viewing and printing with Adobe Acrobat Reader.

1.3.9. Obtaining License Use Information

Your installation of Fluent is managed by the ANSYS License Manager (ANSLIC_ADMIN). To learn more about ANSYS licensing information, refer to the ANSYS, Inc. Licensing Guide within the help viewer.

1.3.10. Version and Release Information

You can obtain information about the version and release of Fluent you are running by selecting the **Version...** option in help menu (accessed by clicking).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 2: Text User Interface (TUI)

In addition to the graphical user interface described in Graphical User Interface (GUI) (p. 1), the user interface in ANSYS Fluent includes a textual command line interface.

2.1. Text Menu System
2.2. Text Prompt System
2.3. Interrupts
2.4. System Commands
2.5. Text Menu Input from Character Strings
2.6. Using the Text Interface Help System

The text interface (TUI) uses, and is written in, a dialect of Lisp called Scheme. Users familiar with Scheme will be able to use the interpretive capabilities of the interface to create customized commands.

2.1. Text Menu System

The text menu system provides a hierarchical interface to the program's underlying procedural interface. Because it is text based, you can easily manipulate its operation with standard text-based tools: input can be saved in files, modified with text editors, and read back in to be executed. Because the text menu system is tightly integrated with the Scheme extension language, it can easily be programmed to provide sophisticated control and customized functionality.

The menu system structure is similar to the directory tree structure of Linux operating systems. When you first start ANSYS Fluent, you are in the "root" menu and the menu prompt is a greater-than symbol.

```
>
```

To generate a listing of the submenus and commands in the current menu, simply press **Enter**. Note that the available submenus and commands will depend on whether you are in meshing mode or solution mode. The submenus and commands that are available from the root menu of the solution mode are as follows:

```
>Enter
adapt/                  mesh/                   surface/
define/                 parallel/               switch-to-meshing-mode
display/                plot/                   views/
exit                    report/
file/                   solve/
```

Note that `switch-to-meshing-mode` is only available for 3D sessions, before you have read a mesh or a case file.

By convention, submenu names end with a `/` to differentiate them from menu commands. To execute a command, just type its name (or an abbreviation). Similarly, to move down into a submenu, enter its name or an abbreviation. When you move into the submenu, the prompt will change to reflect the current menu name.

```
>display

/display> set

/display/set>
```

To move back to the previously occupied menu, type `q` or `quit` at the prompt.

```
/display/set> q

/display>
```

You can move directly to a menu by giving its full pathname.

```
/display> /file

/display//file>
```

In the above example, control was passed from `/display` to `/file` without stopping in the root menu. Therefore, when you quit from the `/file` menu, control will be passed directly back to `/display`.

```
/display//file> q

/display>
```

Furthermore, if you execute a command without stopping in any of the menus along the way, control will again be returned to the menu from which you invoked the command.

```
/display> /file start-journal jrnl

 Opening input journal to file "jrnl".

/display>
```

The text menu system provides online help for menu commands. The text menu online help system is described in Using the Text Interface Help System (p. 36).

To edit the current command, you can position the cursor with the left and right arrow keys, delete with the **Backspace** key, and insert text simply by typing.

For additional information, see the following sections:
2.1.1. Command Abbreviation
2.1.2. Command Line History
2.1.3. Scheme Evaluation
2.1.4. Aliases

2.1.1. Command Abbreviation

To select a menu command, you do not need to type the entire name; you can type an abbreviation that matches the command. The rules for matching a command are as follows: A command name consists of phrases separated by hyphens. A command is matched by matching an initial sequence of its phrases. Matching of hyphens is optional. A phrase is matched by matching an initial sequence of its characters. A character is matched by typing that character.

If an abbreviation matches more than one command, then the command with the greatest number of matched phrases is chosen. If more than one command has the same number of matched phrases, then the command is chosen based on the internal ordering of the commands in the code. Confirm that any command abbreviation calls the command you expect before proceeding.

For example, each of the following will match the command `set-ambient-color`: `set- ambient-color`, `s-a-c`, `sac`, and `sa`. When abbreviating commands, sometimes your abbreviation will match more than one command. In such cases, the first command is selected. Occasionally, there is an anomaly such as `lint` not matching `lighting- interpolation` because the `li` gets absorbed in `lights-`

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

on? and then the `nt` does not match `interpolation`. This can be resolved by choosing a different abbreviation, such as `liin`, or `l-int`.

Important

It is not recommended to use abbreviations in journals/scripts as it can make it difficult for anyone reviewing the journal/script to determine the commands being entered.

2.1.2. Command Line History

You can use the up and down arrow keys on your keyboard to go through recently used commands that are stored in history. By default, command-history will store only the last ten commands. This can be changed (for example to 15) by using the following command:

```
> (set! *cmd-history-length* 15)
```

Important

Command-history is not available if the ANSYS Fluent application is started with -g options (see Command Line Startup Options in the Getting Started Guide).

Important

The user inputs supplied as the arguments of the TUI command or alias will not be saved in history. By way of illustration, consider the following entry in the TUI:

```
> rc new_file.cas
```

Important

In history, only `rc` (an alias for `read-case`) will be saved, since `new_file.cas` is a user input to the alias-function.

Commands recalled from history can be edited or corrected using the **Backspace** key and the left and right arrow keys.

2.1.3. Scheme Evaluation

If you enter an open parenthesis, `(`, at the menu prompt, then that parenthesis and all characters up to and including the matching closing parenthesis are passed to Scheme to be evaluated, and the result of evaluating the expression is displayed.

```
> (define a 1)

a

> (+ a 2 3 4)

10
```

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.1.4. Aliases

Command aliases can be defined within the menu system. As with the Linux `csh` shell, aliases take precedence over command execution. The following aliases are predefined in Cortex: `error`, `pwd`, `chdir`, `ls`, `.`, and `alias`.

`error`
displays the Scheme object that was the "irritant" in the most recent Scheme error interrupt.

`pwd`
prints the working directory in which all file operations will take place.

`chdir`
will change the working directory.

`ls`
lists the files in the working directory.

`.` (period)
prompts you for a journal file name and then run the journal file you have specified.

`alias`
displays the list of symbols currently aliased.

2.2. Text Prompt System

Commands require various arguments, including numbers, filenames, yes/no responses, character strings, and lists. A uniform interface to this input is provided by the text prompt system. A prompt consists of a prompt string, followed by an optional units string enclosed in parentheses, followed by a default value enclosed in square brackets. The following shows some examples of prompts:

```
filled-mesh? [no] Enter

shrink-factor [0.1] Enter

line-weight [1] Enter

title [""]Enter
```

The default value for a prompt is accepted by pressing **Enter** on the keyboard or typing a `,` (comma).

Important

Note that a comma is not a separator. It is a separate token that indicates a default value. The sequence "`1,2`" results in three values; the number 1 for the first prompt, the default value for the second prompt, and the number 2 for the third prompt.

A short help message can be displayed at any prompt by entering a ?. (See Using the Text Interface Help System (p. 36).)

To abort a prompt sequence, simply press **Ctrl**+**c**.

For additional information, see the following sections:
2.2.1. Numbers
2.2.2. Booleans

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.2.3. Strings
2.2.4. Symbols
2.2.5. Filenames
2.2.6. Lists
2.2.7. Evaluation
2.2.8. Default Value Binding

2.2.1. Numbers

The most common prompt type is a number. Numbers can be either integers or reals. Valid numbers are, for example, `16`, `-2.4`, `.9E5`, and `+1E-5`. Integers can also be specified in binary, octal, and hexadecimal form. The decimal integer `31` can be entered as `31`, `#b11111`, `#o37`, or `#x1f`. In Scheme, integers are a subset of reals, so you do not need a decimal point to indicate that a number is real; `2` is just as much a real as `2.0`. If you enter a real number at an integer prompt, any fractional part will simply be truncated; `1.9` will become `1`.

2.2.2. Booleans

Some prompts require a yes-or-no response. A yes/no prompt will accept either `yes` or `y` for a positive response, and `no` or `n` for a negative response. Yes/no prompts are used for confirming potentially dangerous actions such as overwriting an existing file, exiting without saving case, data, mesh, and so on.

Some prompts require actual Scheme Boolean values (true or false). These are entered with the Scheme symbols for true and false, `#t` and `#f`.

2.2.3. Strings

Character strings are entered in double quotes, for example, `"red"`. Plot titles and plot legend titles are examples of character strings. Character strings can include any characters, including blank spaces and punctuation.

2.2.4. Symbols

Symbols are entered *without* quotes. Zone names, surface names, and material names are examples of symbols. Symbols must start with an alphabetical character (that is, a letter), and cannot include any blank spaces or commas.

2.2.5. Filenames

Filenames are actually just character strings. For convenience, filename prompts do not require the string to be surrounded with double quotes. If, for some exceptional reason, a filename contains an embedded space character, then the name must be surrounded with double quotes.

One consequence of this convenience is that filename prompts do not evaluate the response. For example, the sequence

```
> (define fn "valve.ps")

fn

> hc fn
```

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

will end up writing a picture file with the name `fn`, not `valve.ps`. Since the filename prompt did not evaluate the response, `fn` did not get a chance to evaluate `"valve.ps"` as it would for most other prompts.

2.2.6. Lists

Some functions in ANSYS Fluent require a "list" of objects such as numbers, strings, Booleans, and so on. A list is a Scheme object that is simply a sequence of objects terminated by the empty list, `'()`. Lists are prompted for an element at a time, and the end of the list is signaled by entering an empty list. This terminating list forms the tail of the prompted list, and can either be empty or can contain values. For convenience, the empty list can be entered as `()` as well as the standard form `'()`. Normally, list prompts save the previous argument list as the default. To modify the list, overwrite the desired elements and terminate the process with an empty list. For example,

```
element(1) [()] 1

element(2) [()] 10

element(3) [()] 100

element(4) [()] Enter
```

creates a list of three numbers: 1, 10, and 100. Subsequently,

```
element(1) [1] Enter

element(2) [10] Enter

element(3) [100] Enter

element(4) [()] 1000

element(5) [()] Enter
```

adds a fourth element. Then

```
element(1) [1] Enter

element(2) [10] Enter

element(3) [100] ()
```

leaves only 1 and 10 in the list. Subsequently entering

```
element(1) [1] ,,'(11 12 13)
```

creates a five element list: 1, 10, 11, 12, and 13. Finally, a single empty list removes all elements

```
element(1) [1] ()
```

A different type of list, namely, a "list-of-scalars" contains pick menu items (and not list items) for which a selection has to be made from listed quantities, which are available at the **Enter** prompt. Hence, a list-of-scalars cannot be entered as a list.

An example of a "list-of-scalars" consists of the following:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
ASCII scalar(1)>
abs-angular-coordinate  cell-warp              radial-velocity
absolute-pressure       cell-weight            rel-tangential-velocity
adaption-curvature      cell-zone              rel-total-pressure
adaption-function       custom-function-0      rel-velocity-magnitude
adaption-iso-value      density                relative-velocity-angle
adaption-space-gradient density-all            relative-x-velocity
angular-coordinate      dp-dx                  relative-y-velocity
axial-coordinate        dp-dy                  relative-z-velocity
axial-velocity          dp-dz                  tangential-velocity
boundary-cell-dist      dynamic-pressure       total-pressure
boundary-normal-dist    existing-value         velocity-angle
boundary-volume-dist    face-area-magnitude    velocity-magnitude
cell-children           face-handedness        x-coordinate
cell-element-type       face-squish-index      x-face-area
cell-equiangle-skew     interface-wall-zone    x-velocity
cell-equivolume-skew    mass-imbalance         x-velocity-residual
cell-partition          mesh-x-velocity        y-coordinate
cell-refine-level       mesh-y-velocity        y-face-area
cell-squish-index       mesh-z-velocity        y-velocity
cell-surface-area       partition-neighbors    y-velocity-residual
cell-type               pressure               z-coordinate
cell-volume             pressure-coefficient   z-face-area
cell-volume-change      pressure-residual      z-velocity
cell-wall-distance      radial-coordinate      z-velocity-residual

ASCII scalar(1)> pressure

ASCII scalar(2)> z-coordinate

ASCII scalar(3)> cell-volume

ASCII scalar(4)> dynamic-pressure velocity-magnitude x-face-area

ASCII scalar(7)>
```

2.2.7. Evaluation

All responses to prompts (except filenames, see above) are evaluated by the Scheme interpreter before they are used. You can therefore enter any valid Scheme expression as the response to a prompt. For example, to enter a unit vector with one component equal to $1/3$ (without using your calculator),

```
/foo> set-xy
x-component [1.0] (/ 1 3)

y-component [0.0] (sqrt (/ 8 9))
```

or, you could first define a utility function to compute the second component of a unit vector,

```
> (define (unit-y x) (sqrt (- 1.0 (* x x))))

unit-y
/foo> set-xy
```

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
x-component [1.0] (/ 1 3)

y-component [0.0] (unit-y (/ 1 3))
```

2.2.8. Default Value Binding

The default value at any prompt is bound to the Scheme symbol "_" (underscore) so that the default value can form part of a Scheme expression. For example, if you want to decrease a default value so that it is one-third of the original value, you could enter

```
shrink-factor [0.8] (/ _ 3)
```

2.3. Interrupts

The execution of the code can be halted by pressing the **Ctrl**+**c**, at which time the present operation stops at the next recoverable location.

2.4. System Commands

The way you execute system commands with the ! (bang) shell escape character will be slightly different for Linux and Windows systems.

For additional information, see the following sections:
2.4.1. System Commands for Linux-based Operating Systems
2.4.2. System Commands for Windows Operating Systems

2.4.1. System Commands for Linux-based Operating Systems

If you are running ANSYS Fluent under a Linux-based operating system, all characters following the ! up to the next newline character will be executed in a subshell. Any further input related to these system commands must be entered in the window in which you started the program, and any screen output will also appear in that window. (Note that if you started ANSYS Fluent remotely, this input and output will be in the window in which you started Cortex.)

```
> !rm junk.*

> !vi script.rp
```

`!pwd` and `!ls` will execute the Linux commands in the directory in which Cortex was started. The screen output will appear in the window in which you started ANSYS Fluent, unless you started it remotely, in which case the output will appear in the window in which you started Cortex. (Note that `!cd` executes in a subshell, so it will not change the working directory either for ANSYS Fluent or for Cortex, and is therefore not useful.) Typing `cd` with no arguments will move you to your home directory in the console.

ANSYS Fluent includes three system command aliases (`pwd`, `ls`, and `chdir`) that will be executed in your working directory with output displayed in the ANSYS Fluent console. Note that these aliases will invoke the corresponding Linux commands with respect to the parent directory of the case file. For example, `pwd` prints the parent directory of the case file in the ANSYS Fluent console, while `!pwd` prints the directory from which you started ANSYS Fluent in the Linux shell window where you started ANSYS Fluent.

Several examples of system commands entered in the console are shown below. The screen output that will appear in the window in which ANSYS Fluent was started (or, if you started the program remotely, in the window in which Cortex was started) follows the examples.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example input (in the ANSYS Fluent console):

```
> !pwd

> !ls valve*.*
```

Example output (in the window in which ANSYS Fluent— or Cortex, if you started the program remotely—was started):

```
/home/cfd/run/valve
 valve1.cas  valve1.msh  valve2.cas  valve2.msh
```

2.4.2. System Commands for Windows Operating Systems

If you are running ANSYS Fluent under a Windows operating system, all characters following the ! up to the next newline character will be executed. The results of a command will appear in the ANSYS Fluent console, or in a separate window if the command starts an external program, such as Notepad.

```
> !del junk.*

> !notepad script.rp
```

`!cd` and `!dir` will execute the DOS commands and the screen output will appear in the ANSYS Fluent console. The `!cd` command with no argument will display the current working directory in the ANSYS Fluent console.

Several examples of system commands entered in the console are shown below.

Example input (in boxes) and output (in the ANSYS Fluent console):

```
  > !cd

p:/cfd/run/valve
 > !dir valve*.*/w

     Volume in drive P is users
     Volume Serial Number is 1234-5678
     Directory of p:/cfd/run/valve
     valve1.cas      valve1.msh      valve2.cas      valve2.msh
        4 File(s)              621,183 bytes
        0 Dir(s)          1,830,088,704 bytes free
```

2.5. Text Menu Input from Character Strings

Often, when writing a Scheme extension function for ANSYS Fluent, it is convenient to be able to include menu commands in the function. This can be done with `ti-menu-load-string`. For example, to open graphics window 2, use

```
(ti-menu-load-string "di ow 2")
```

A Scheme loop that will open windows 1 and 2 and display the front view of the mesh in window 1 and the back view in window 2 is given by

```
(for-each
 (lambda (window view)
   (ti-menu-load-string (format #f "di ow ~a gr view rv ~a"
 window view)))
 '(1 2)
 '(front back))
```

This loop makes use of the `format` function to construct the string used by `menu-load-string`. This simple loop could also be written without using menu commands at all, but you need to know the Scheme functions that get executed by the menu commands to do it:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
(for-each
 (lambda (window view)
   (cx-open-window window)
   (display-mesh)
   (cx-restore-view view))
 '(1 2) '(front back))
```

String input can also provide an easy way to create aliases within ANSYS Fluent. For example, to create an alias that will display the mesh, you could type the following:

```
(alias 'dg (lambda () (ti-menu-load-string "/di gr")))
```

Then any time you enter `dg` from anywhere in the menu hierarchy, the mesh will be drawn in the active window.

Important

`ti-menu-load-string` evaluates the string argument in the top level menu. It ignores any menu you may be in when you invoke `ti-menu-load-string`.

As a result, the command

```
(ti-menu-load-string "open-window 2 gr") ; incorrect usage
```

will not work even if you type it from within the `display/` menu—the string itself must cause control to enter the `display/` menu, as in

```
(ti-menu-load-string "display open-window 2 mesh")
```

2.6. Using the Text Interface Help System

The text user interface provides context-sensitive online help. Within the text menu system, you can obtain a brief description of each of the commands by entering a ? followed by the command in question.

Example:

```
> ?dis
display/: Enter the display menu.
```

You can also enter a lone ? to enter "help mode." In this mode, you need only enter the command or menu name to display the help message. To exit the help mode type `q` or `quit` as for a normal menu.

Example:

```
> ?

[help-mode]> di

display/: Enter the display menu.

 [help-mode]> pwd

 pwd: #[alias]
 (LAMBDA ()
   (BEGIN
   (SET! pwd-cmd ((LAMBDA n
  n) 'system (IF (cx-send '(unix?))
    "pwd"
    "cd")))
  (cx-send pwd-cmd)))
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
[help-mode]> q
```

To access the help, type a ? at the prompt when you are prompted for information.

Example:

```
> display/annotate Annotation text [""] ?
 Annotation text [""]
```

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 3: Reading and Writing Files

During an ANSYS Fluent session you may need to import and export several kinds of files. Files that are read include mesh, case, data, profile, Scheme, and journal files. Files that are written include case, data, profile, journal, and transcript files. ANSYS Fluent also has features that enable you to save pictures of graphics windows. You can also export data for use with various visualization and postprocessing tools. These operations are described in the following sections.

3.1. Shortcuts for Reading and Writing Files
3.2. Reading Mesh Files
3.3. Reading and Writing Case and Data Files
3.4. Reading and Writing Parallel Data Files
3.5. Reading Fluent/UNS and RAMPANT Case and Data Files
3.6. Reading and Writing Profile Files
3.7. Reading and Writing Boundary Conditions
3.8. Writing a Boundary Mesh
3.9. Reading Scheme Source Files
3.10. Creating and Reading Journal Files
3.11. Creating Transcript Files
3.12. Importing Files
3.13. Exporting Solution Data
3.14. Exporting Solution Data after a Calculation
3.15. Exporting Steady-State Particle History Data
3.16. Exporting Data During a Transient Calculation
3.17. Exporting to ANSYS CFD-Post
3.18. Managing Solution Files
3.19. Mesh-to-Mesh Solution Interpolation
3.20. Mapping Data for Fluid-Structure Interaction (FSI) Applications
3.21. Saving Picture Files
3.22. Setting Data File Quantities
3.23. The .fluent File

3.1. Shortcuts for Reading and Writing Files

The following features in ANSYS Fluent make reading and writing files convenient:

- Automatic appending or detection of default file name suffixes
- Binary file reading and writing
- Automatic detection of file format (text/binary)
- Recent file list
- Reading and writing of compressed files
- Tilde expansion
- Automatic numbering of files

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Ability to disable the overwrite confirmation prompt
- Standard toolbar buttons for reading and writing files

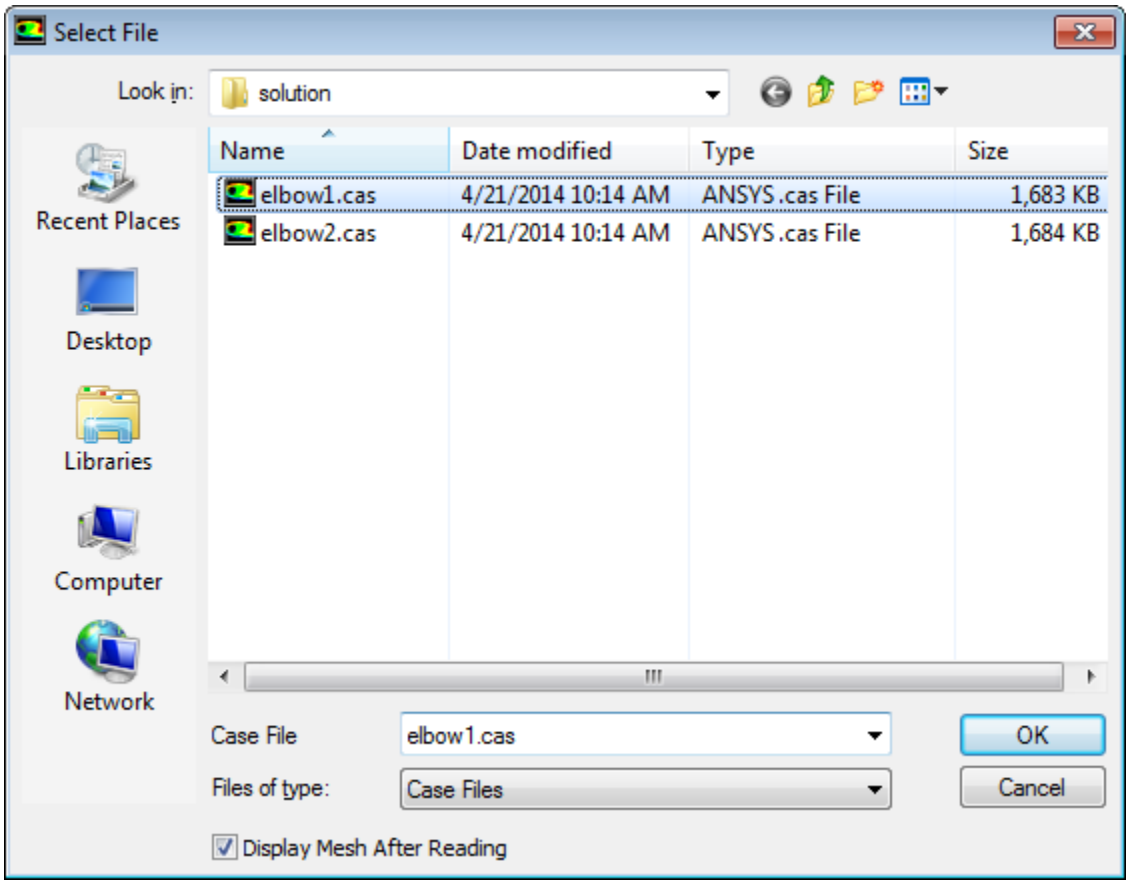
For additional information, see the following sections:

3.1.1. Default File Suffixes
3.1.2. Binary Files
3.1.3. Detecting File Format
3.1.4. Recent File List
3.1.5. Reading and Writing Compressed Files
3.1.6. Tilde Expansion (Linux Systems Only)
3.1.7. Automatic Numbering of Files
3.1.8. Disabling the Overwrite Confirmation Prompt
3.1.9. Toolbar Buttons

3.1.1. Default File Suffixes

Each type of file read or written in ANSYS Fluent has a default file suffix associated with it. When you specify the first part of the file name (the prefix) for the commonly used files, Fluent automatically appends or detects the appropriate suffix. For example, to write a case file named `myfile.cas`, just specify the prefix `myfile` in the **Select File** dialog box (Figure 3.1: The Select File Dialog Box (p. 40)) and `.cas` is automatically appended. Similarly, to read the case file named `myfile.cas` into Fluent, you can just specify `myfile` and ANSYS Fluent automatically searches for a file of that name with the suffix `.cas`.

Figure 3.1: The Select File Dialog Box



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The default file suffix for case and data files, PDF (Probability Density Function) files, DTRM ray files, profiles, scheme files, journal files, and so on, are automatically detected and appended. The appropriate default file suffix appears in the **Select File** dialog box for each type of file.

3.1.2. Binary Files

When you write a case, data, or ray file, a binary file is saved by default. Binary files take up less memory than text files and can be read and written by ANSYS Fluent more quickly.

Note

You cannot read and edit a binary file, as you can do for a text file.

To save a text file, turn off the **Write Binary Files** option in the **Select File** dialog box when you are writing the file. Note that if you use the HDF file format to write out your case and data files, they will always be binary (Reading and Writing Files Using Hierarchical Data Format (HDF) (p. 47)).

3.1.3. Detecting File Format

When you read a case, data, mesh, PDF, or ray file, Fluent automatically determines whether it is a text (formatted) or binary file.

3.1.4. Recent File List

At the bottom of the **File/Read** ribbon tab submenu there is a list of four ANSYS Fluent case files that you most recently read or wrote. To read one of these files into ANSYS Fluent, select it in the list. This allows you to read a recently used file without selecting it in the **Select File** dialog box.

Note that the files listed in this submenu may not be appropriate for your current session (for example, a 3D case file can be listed even if you are running a 2D version of ANSYS Fluent). Also, if you read a case file using this shortcut, the corresponding data file is read only if it has the same base name as the case file (for example, `file1.cas` and `file1.dat`) *and* it was read/written with the case file the last time the case file was read/written.

3.1.5. Reading and Writing Compressed Files

For more information, see the following sections:

3.1.5.1. Reading Compressed Files
3.1.5.2. Writing Compressed Files

3.1.5.1. Reading Compressed Files

You can use the **Select File** dialog box to read files compressed using `compress` or `gzip`. If you select a compressed file with a `.Z` extension, ANSYS Fluent automatically opens `zcat` to import the file. If you select a compressed file with a `.gz` extension, Fluent opens `gunzip` to import the file.

For example, if you select a file named `flow.msh.gz`, Fluent reports the following message indicating that the result of the `gunzip` is imported into ANSYS Fluent via an operating system pipe.

```
Reading "\"| gunzip -c \"Y:\flow.msh.gz\"\""...
```

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also type in the file name without any suffix (for example, if you are not sure whether or not the file is compressed). First, Fluent attempts to open a file with the input name. If it cannot find a file with that name, it attempts to locate files with default suffixes and extensions appended to the name.

For example, if you enter the name `file-name`, Fluent traverses the following list until it finds an existing file:

- `file-name`
- `file-name.gz`
- `file-name.Z`
- `file-name.` *suffix*
- `file-name.` *suffix* `.gz`
- `file-name.` *suffix* `.Z`

where *suffix* is a common extension to the file, such as `.cas` or `.msh`. Fluent reports an error if it fails to find an existing file with one of these names.

Note

In addition to `.gz` and `.Z` compression, ANSYS Fluent can also handle `.bz2` compressed files.

Important

- For Windows systems, only files that were compressed with `gzip` (that is, files with a `.gz` extension) can be read. Files that were compressed with `compress` cannot be read into ANSYS Fluent on a Windows machine.
- Do not read a compressed ray file; ANSYS Fluent cannot access the ray tracing information properly from a compressed ray file.

3.1.5.2. Writing Compressed Files

You can use the **Select File** dialog box to write a compressed file by appending a `.Z` or `.gz` extension onto the file name.

For example, if you enter `flow.gz` as the name for a case file, Fluent reports the following message:

```
Writing "| gzip -cfv > Y:\flow.cas.gz"...
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The status message indicates that the case file information is being piped into the `gzip` command, and that the output of the compression command is being redirected to the file with the specified name. In this particular example, the `.cas` extension is added automatically.

Note

In addition to `.gz` and `.Z` compression, ANSYS Fluent can also handle `.bz2` compressed files.

Important

- For Windows systems, compression can be performed only with `gzip`. That is, you can write a compressed file by appending `.gz` to the name, but appending `.Z` does not compress the file.
- Do not write a compressed ray file; ANSYS Fluent cannot access the ray tracing information properly from a compressed ray file.
- To compress parallel data files (that is, files saved with a `.pdat` extension), you must use asynchronous file compression. (For details, see Reading and Writing Case and Data Files (p. 45)).

3.1.6. Tilde Expansion (Linux Systems Only)

On Linux systems, if you specify `~/` as the first two characters of a file name, the `~` is expanded as your home directory. Similarly, you can start a file name with `~username/`, and the `~username` is expanded to the home directory of "username". If you specify `~/file` as the case file to be written, ANSYS Fluent saves the file `file.cas` in your home directory. You can specify a subdirectory of your home directory as well: if you enter `~/cases/file.cas`, ANSYS Fluent saves the file `file.cas` in the `cases` subdirectory.

3.1.7. Automatic Numbering of Files

There are several special characters that you can include in a file name. Using one of these character strings in your file name provides a shortcut for numbering the files based on various parameters (that is, iteration number, time step, or total number of files saved so far), because you need not enter a new file name each time you save a file. (See also Automatic Saving of Case and Data Files (p. 48) for information about saving and numbering case and data files automatically.)

- For transient calculations, you can save files with names that reflect the time step at which they are saved by including the character string `%t` in the file name. For example, you can specify `contours-%t.ps` for the file name, and Fluent saves a file with the appropriate name (for example, `contours-0001.ps` if the solution is at the first time step).

  This automatic saving of files with the time step should not be used for steady-state cases, since the time step will always remain zero.

- For transient calculations, you can save files with names that reflect the flow-time at which they are saved by including the character string `%f` in the file name. The usage is similar to `%t`. For example, when you specify `filename-%f.ps` for the file name, Fluent will save a file with the appropriate name (for example, `filename-005.000000.ps` for a solution at a flow-time of 5 seconds). By default, the flow-time that is included in the file name will have a field width of 10 and 6 decimal places. To modify this format, use the character string `%`*x*`.`*y*`f`, where *x* and *y* are the preferred field width and number of decimal places, respect-

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ively. ANSYS Fluent will automatically add zeros to the beginning of the flow-time to achieve the prescribed field width. To eliminate these zeros and left align the flow-time, use the character string `%- x . y f` instead.

This automatic saving of files with flow-time should not be used for steady-state cases, since the flow-time will always remain zero.

- To save a file with a name that reflects the iteration at which it is saved, use the character string `%i` in the file name. For example, you can specify `contours-%i.ps` for the file name, and Fluent saves a file with the appropriate name (for example, `contours-0010.ps` if the solution is at the 10th iteration).
- To save a picture file with a name that reflects the total number of picture files saved so far in the current Fluent session, use the character string %n in the file name. This option can be used only for picture files.

The default field width for `%i`, `%t`, and %n formats is 4. You can change the field width by using % *x* `i`, % *x* `t`, and % *x* `n` in the file name, where *x* is the preferred field width.

3.1.8. Disabling the Overwrite Confirmation Prompt

By default, if you ask ANSYS Fluent to write a file with the same name as an existing file in that folder, it will ask you to confirm that it is "OK to overwrite" the existing file. If you do not want Fluent to ask you for confirmation before it overwrites existing files, you can use the **Batch Options** dialog box (see Batch Execution Options in the Getting Started Guide for details). Alternatively, enter the `file/confirm-overwrite?` text command and answer `no` (see Text User Interface (TUI) (p. 27) for the text user interface commands).

3.1.9. Toolbar Buttons

The standard toolbar provides buttons that make it easier to read and write files:

- The **Read a file** button ( ) allows you to read existing files using a file selection dialog box. The files available for reading include all those available through the **File/Read** ribbon tab item, as described in this chapter.
- The **Write a file** button ( ) allows you to write various types of files. The files available for writing include all those available through the **File/Write** ribbon tab item, as described in this chapter.

3.2. Reading Mesh Files

Mesh files are created using the mesh generators (ANSYS Meshing, the meshing mode of Fluent, Fluent Meshing, GAMBIT, GeoMesh, and PreBFC), or by several third-party CAD packages. From the point of view of ANSYS Fluent, a mesh file is a subset of a case file (described in Reading and Writing Case Files (p. 46)). The mesh file contains the coordinates of all the nodes, connectivity information that tells how the nodes are connected to one another to form faces and cells, and the zone types and numbers of all the faces (for example, wall-1, pressure-inlet-5, symmetry-2).

The mesh file does not contain any information on boundary conditions, flow parameters. For information about meshes, see Reading and Manipulating Meshes (p. 115).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To read a native-format mesh file (that is, a mesh file that is saved in the ANSYS Fluent format, which includes FLUENT 5/6, Fluent/UNS, or RAMPANT meshes) into Fluent, use the **File/Read/Mesh...** ribbon tab item.

File → **Read** → **Mesh...**

Note that you can also use the **File/Read/Case...** ribbon tab item (described in Reading and Writing Case Files (p. 46)), because a mesh file is a subset of a case file. ANSYS Meshing, the meshing mode of Fluent, Fluent Meshing, GAMBIT, GeoMesh, and PreBFC can all write a native-format mesh file. For information about these files and how they are created, see ANSYS Meshing Mesh Files (p. 135), Fluent Meshing Mode Mesh Files (p. 136), Fluent Meshing Mesh Files (p. 136), GAMBIT Mesh Files (p. 136), GeoMesh Mesh Files (p. 136), and PreBFC Mesh Files (p. 136).

If after reading in a mesh file (or a case and data file), you would like to read in another mesh file, the Read Mesh Options Dialog Box (p. 2634) will open, where you can choose to

- Discard the case and read in a new mesh.
- Replace the existing mesh.

You also have the option to have the **Scale Mesh** dialog box appear automatically for you to check or scale your mesh, which in general is the recommended practice. For this to happen, enable **Show Scale Mesh Panel After Replacing Mesh**.

For information on importing an unpartitioned mesh file into the parallel version of Fluent using the partition filter, see Using the Partition Filter (p. 2052).

For information about reading surface mesh files, see Reading Surface Mesh Files (p. 149).

3.3. Reading and Writing Case and Data Files

Information related to the ANSYS Fluent simulation is stored in both the case file and the data file. The commands for reading and writing these files are described in the following sections, along with commands for the automatic saving of case and data at specified intervals.

ANSYS Fluent can read and write either text or binary case and data files. Binary files require less storage space and are faster to read and write. By default, ANSYS Fluent writes files in binary format. To write a text file, disable the **Write Binary Files** check button in the **Select File** dialog box. In addition, you can read and write either text or binary files in compressed formats (For details, see Reading and Writing Compressed Files (p. 41)). ANSYS Fluent automatically detects the file type when reading.

Furthermore, when writing case and data files, ANSYS Fluent can improve I/O performance using asynchronous file compression which can be enabled using the **Optimize Using Asynchronous I/O** check button in the **Select File** dialog box.

This option is particularly beneficial if you want to improve the overall performance of simulations involving solution check-pointing. Performance is improved by overlapping the file compression and file system copy operations with subsequent iterations or other operations. It is best if the front end Cortex

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

process has its own dedicated processor core (or machine) not used by the ANSYS Fluent compute-node processes, even though the option would aid performance even if they reside on the same machine.

Important

- The **Optimize Using Asynchronous I/O** option is not available on Windows machines.
- If you adapt the mesh, you must save a new case file as well as a data file. Otherwise, the new data file will not correspond to the case file (for example, they will have different numbers of cells). If you have not saved the latest case or data file, ANSYS Fluent will warn you when you try to exit the program.

For additional information, see the following sections:

3.3.1. Reading and Writing Case Files
3.3.2. Reading and Writing Data Files
3.3.3. Reading and Writing Case and Data Files Together
3.3.4. Reading and Writing Files Using Hierarchical Data Format (HDF)
3.3.5. Automatic Saving of Case and Data Files

3.3.1. Reading and Writing Case Files

Case files contain the mesh, boundary and cell zone conditions, and solution parameters for a problem. It also contains the information about the user interface and graphics environment. For information about the format of case files see Case and Data File Formats (p. 2851). The commands used for reading case files can also be used to read native-format mesh files (as described in Reading Mesh Files (p. 44)) because the mesh information is a subset of the case information. Select the **File/Read/Case...** ribbon tab item to open the **Select File** dialog box.

File → **Read** → **Case...**

Read a case file using the **Select File** dialog box. Note that the **Display Mesh After Reading** option in the **Select File** dialog box allows you to have the mesh displayed automatically after it is read.

Select the **File/Write/Case...** ribbon tab item to open the **Select File** dialog box.

File → **Write** → **Case...**

Write a case file using the **Select File** dialog box.

When ANSYS Fluent reads a case file, it first looks for a file with the exact name you typed. If a file with that name is not found, it searches for the same file with different extensions (Reading and Writing Compressed Files (p. 41)). When ANSYS Fluent writes a case file, `.cas` is added to the name you type unless the name already ends with `.cas`.

In the single-precision version of ANSYS Fluent, nodal coordinates are written by default in double precision, in order to ensure that there is no loss of precision if the case file is read back into the meshing mode. You can reduce the precision and case file size by using the following text command prior to writing:

```
file → single-precision-coordinates?
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.3.2. Reading and Writing Data Files

Data files contain the values of the specified flow field quantities in each mesh element and the convergence history (residuals) for that flow field. For information about the format of data files see Case and Data File Formats (p. 2851).

After reading a mesh or case file, select the **File/Read/Data...** ribbon tab item to open the **Select File** dialog box.

File → **Read** → **Data...**

Read a data file using the **Select File** dialog box.

After generating data for a case file, select the **File/Write/Data...** ribbon tab item to open the **Select File** dialog box.

File → **Write** → **Data...**

Write a data file using the **Select File** dialog box.

When ANSYS Fluent reads a data file, it first looks for a file with the exact name you typed. If a file with that name is not found, it searches for the same file with different extensions (Reading and Writing Compressed Files (p. 41)). When ANSYS Fluent writes a data file, `.dat` is added to the name you type unless the name already ends with `.dat`.

3.3.3. Reading and Writing Case and Data Files Together

A case file and a data file together contain all the information required to restart a solution. Case files contain the mesh, boundary and cell zone conditions, and solution parameters. Data files contain the values of the flow field in each mesh element and the convergence history (residuals) for that flow field.

You can read a case file and a data file together by using the **Select File** dialog box opened by selecting the **File/Read/Case & Data...** ribbon tab item. To read both files, select the appropriate case file, and the corresponding data file (same name with `.dat` suffix) is also read. Note that the **Display Mesh After Reading** option in the **Select File** dialog box allows you to have the mesh displayed automatically after it is read.

File → **Read** → **Case & Data...**

To write a case file and a data file, select the **File/Write/Case & Data...** ribbon tab item.

File → **Write** → **Case & Data...**

3.3.4. Reading and Writing Files Using Hierarchical Data Format (HDF)

When reading and writing case and data files in parallel you can optionally use the Hierarchical Data Format (HDF). To read or write case/data files using HDF, you can use the same GUI or TUI commands as you normally would and simply append `.h5` to the file name. Alternatively, you can select **HDF5 Case Files** or **HDF5 Data Files** from the **Files of type** drop-down list in the **Select File** dialog box. If you want to write HDF files automatically at specified intervals during a calculation, do not append `.h5` to the file name in the Autosave Dialog Box (p. 2527), but instead set HDF as the default for reading and writing files by using the following text command:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`file → hdf-files?`

Note

HDF files are not supported in Fluent Meshing or when running Fluent under ANSYS Workbench.

HDF files are always binary and make use of built-in compression. Thus, they cannot be viewed in a text editor. However, third-party tools are available that allow you to open and explore the contents of files saved in HDF format.

You can specify the I/O mode and/or the HDF compression level through text commands:

```
> file hdfio-options compression-level
compression-level [0-9]? [1] 4

>  file hdfio-options io-mode
HDF5 I/O Modes:
 1. HOST
 2. NODE0
 3. PARALLEL INDEPENDENT
 4. PARALLEL COLLECTIVE
Enter Mode Number: [2] 1

HOST mode selected.
```

The compression level can be set from 0 (no compression; fastest) to 9 (maximum compression; slowest); the default level is 1. Note that increasing the compression level may not necessarily yield a corresponding decrease in file size.

The available I/O modes are:

HOST: I/O is done serially by the host process. This is the default mode when the file path is not accessible to compute-node 0.
NODE0: I/O is done serially by the node 0 process. This is the default mode when the file path is accessible to compute-node 0.
PARALLEL INDEPENDENT: I/O is done in parallel using the independent mode of MPI I/O.
PARALLEL COLLECTIVE: I/O is done in parallel using the collective mode of MPI I/O.

Note

- Reading and Writing files using HDF is only supported in parallel Fluent sessions. If you need to read or write HDF files in a single-process Fluent session, you must start it as a parallel session using a single process, rather than as a serial session.
- The Parallel I/O modes do not support compression. If you have specified a non-zero level for compression and you switch to one of the parallel modes, the compression will be set to 0.

3.3.5. Automatic Saving of Case and Data Files

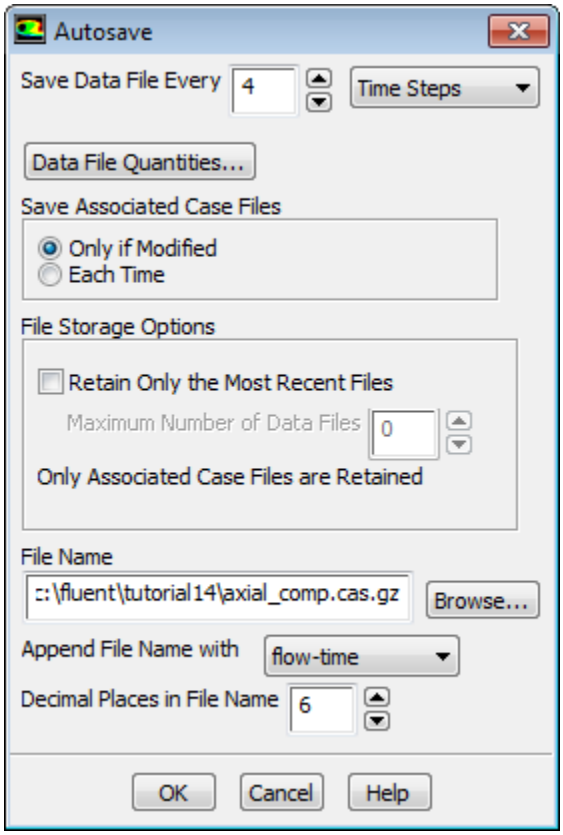
You can request ANSYS Fluent to automatically save case and data files at specified intervals during a calculation. This is especially useful for time-dependent calculations, since it allows you to save the results at different time steps, flow times, or crank angles (for in-cylinder simulations) without stopping the calculation and performing the save manually. You can also use the autosave feature for steady-state problems, and thus examine the solution at different stages in the iteration history.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Automatic saving is specified using the Autosave Dialog Box (p. 2527) (Figure 3.2: The Autosave Dialog Box (p. 49)), which is opened by clicking the **Edit...** button next to the **Autosave Every** text box in the **Calculation Activities** task page.

Calculation Activities

Figure 3.2: The Autosave Dialog Box



Specify how often you would like to save your modified files by entering the frequency in the **Save Data File Every** number-entry field. **Save Data File Every** is set to zero by default, indicating that no automatic saving is performed.

For transient in-cylinder simulations, specify whether you want to save the data by **Time Steps** or **Crank Angles**.

If you choose to save the case file only if it is modified, then select **If Modified During the Calculation or Manually** under **When the Data File is Saved, Save the Case File**. Note that the case file will be saved whether you make a manual change, or if ANSYS Fluent makes a change internally during the calculation. If you choose to save the case file every time the data file is saved, then select **Each Time**.

Important

To save only the data files, use the following TUI option:

`file` → `auto-save` → `case-frequency` → `if-mesh-is-modified`

This will result in the options in the **When the Data File is Saved, Save the Case** group box being disabled in the **Autosave** dialog box. In essence, this TUI command forces ANSYS

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fluent to the save case file only when the mesh is modified. It does not disable case file saving, but reduces it to an absolute minimum. This is necessary to do so since you cannot read a data file without a case file containing a matching mesh.

For steady-state solutions, specify the frequency in iterations. For transient solutions, specify it in time steps (unless you are using the explicit time stepping formulation, in which case specify the frequency in iterations). For transient in-cylinder solutions, you can choose to specify the frequency in crank angles. If you define a frequency of 5, for example, a case file is saved every 5 iterations, time steps, or crank angles.

If you have limited disk space, restrict the number of files saved by ANSYS Fluent by selecting the **Retain Only the Most Recent Files** option. When selected, enter the **Maximum Number of Data Files** you would like to retain. Note that the case and data files are treated separately with regard to the maximum number of files saved when overwriting. For example, if the value of **Maximum Number of Data Files** is set to five, ANSYS Fluent saves a maximum of five case and five data files, irrespective of the frequency. After the maximum limit of files has been saved, ANSYS Fluent begins overwriting the earliest existing file.

Note

When the **Retain only the Most Recent Files** option is selected, the solution history currently in memory will be discarded and the solution history reset.

If you have generated data (either by initializing the solution or running the calculation) you can view the list of standard quantities that will be written to the data file as a result of the autosave, and even select additional quantities for postprocessing in alternative applications. Click the **Data File Quantities...** button to open the **Data File Quantities** dialog box, and make any necessary selections. For details, see Setting Data File Quantities (p. 107).

Enter a root name for the autosave files in the **File Name** text box. When the files are saved, a number will be appended to this root name to indicate the point at which it was saved during the calculation: for steady-state solutions, this will be the iteration number, whereas for transient solutions it will be either the time step number, flow time, or crank angle (depending on your selection in the step that follows). An extension will also be automatically added to the root name (`.cas` or `.dat`). If the specified **File Name** ends in `.gz` or `.Z`, appropriate file compression is performed. For details about file compression, see Reading and Writing Compressed Files (p. 41). For details on autosaving files in the hierarchical data format (HDF), see Reading and Writing Files Using Hierarchical Data Format (HDF) (p. 47).

For transient calculations, make a selection from the **Append File Name with** drop-down list to indicate whether you want the root file name to be appended with the **time-step**, **flow-time** or **crank-angle** (see Figure 3.2: The Autosave Dialog Box (p. 49)). If you select either of the latter two, you can set the **Decimal Places in File Name** to determine the ultimate width of the file name.

Consider a transient case for which you want to save your case and data files at known time steps. The procedure you would follow is to first set the frequency in the **Save Data File Every** text box. Select **Each Time** if you want both case and data files saved at the same interval. Then enter `my_file` for the **File Name**. Finally, select **time-step** from the **Append File Name with** drop-down list. An example of the resulting files saved would be

```
my_file-0005.cas

my_file-0005.dat
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

indicating that these files were saved at the fifth time step.

You can revise the instructions for the previous example to instead save case and data files at known flow times, by selecting **flow-time** from the **Append File Name with** drop-down list. The default **Decimal Places in File Name** will be six. An example of the resulting files saved would be

```
my_file-0.500000.cas
```

```
my_file-0.500000.dat
```

indicating that these files were saved at a flow time of 0.5 seconds.

For steady-state and transient cases, you have the option of automatically numbering the files (as described in Automatic Numbering of Files (p. 43)), and thereby include further information about when the files were saved. This involves the addition of special characters to the **File Name**. For example, you may want the file names to convey the flow times with their corresponding time steps (transient cases only). Select **time-step** from the **Append File Name with**, and enter a **File Name** that ends with `-%f` to automatically number the files with the flow time. Thus, entering a **File Name** of `filename-%f` could result in a saved case file named `filename-000.500000-0010.cas`. The conventions used in this example can be explained as follows:

- `filename-` is the file name you entered when autosaving your solution.
- `000.500000` is the result of the special character `%f` added to the file name, and is the flow time. This flow time has a field width of ten characters, which allows for six decimal places (as discussed in Automatic Numbering of Files (p. 43)).
- `-0010` is the appended **time-step**, as designated by the selection in the **Append File Name with** drop-down list.
- `.cas` is the file extension automatically added when using the autosave option.

All of the autosave inputs are stored in memory when you click **OK** in the **Autosave** dialog box, and can then be saved with the case file.

3.4. Reading and Writing Parallel Data Files

When performing parallel processing, you have the option of utilizing the parallel input/output (I/O) capability of ANSYS Fluent. This capability permits the writing and reading of data files directly to and from the node processes in a parallel fashion. This is in contrast to the standard ANSYS Fluent I/O approach, in which all data is passed from the node processes to the host process where it is then written out to disk in serial. The motivation for performing the I/O directly from the node processes is to reduce the time for the data file I/O operations.

During parallel I/O operations, all the node processes write to or read from the same file, that is, they concurrently access a single file. The format of this parallel data file is different than the format of the standard ANSYS Fluent data file. The format has been revised to permit more efficient concurrent access. The order of data written to the parallel data file is dependent on the number of processes involved in the session and on the partitioning. However, a parallel data file written from any parallel session on a given platform (that is, any number of processes or partitioning) can be read by any other parallel session on the same platform. In other words, analyses may be freely restarted using a different number of processes, as long as the files are being written or read from the same platform.

For information about parallel processing, see Parallel Processing (p. 2009).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For additional information, see the following sections:

3.4.1. Writing Parallel Data Files
3.4.2. Reading Parallel Data Files
3.4.3. Availability and Limitations

3.4.1. Writing Parallel Data Files

To write a parallel data file, select the **File/Write/Data...** ribbon tab item to open the **Select File** dialog box.

File → **Write** → **Data...**

In the **Select File** dialog box, enter the name of the output data file with a `.pdat` extension. The parallel data files are always binary, so you do not have to enable the **Write Binary Files** option.

Note that parallel data files written from sessions with different numbers of processes may be slightly different in size. However, they contain essentially the same information.

You can also use the text command `file/write-pdat?` to write a `.pdat` file. This is particularly useful if you want to write a `.pdat` file using the autosave data file option, after reading in a data file. The `.pdat` files will be saved during the autosave.

3.4.2. Reading Parallel Data Files

To read parallel data files, select the **File/Read/Data...** ribbon tab item to open the **Select File** dialog box.

File → **Read** → **Data...**

In the **Select File** dialog box, enter the name of the input data file with a `.pdat` extension.

Note that a parallel file will be read slightly quicker into a session that uses the same number of processes as the session that originally wrote the file, as opposed to a session with a different number.

3.4.3. Availability and Limitations

The conditions in which parallel I/O is available and the limitations related to this capability are as follows:

- The parallel I/O capability is only available for parallel ANSYS Fluent sessions. Parallel data files cannot be read into a serial ANSYS Fluent session, although they can be read if ANSYS Fluent is launched with the number of processes set to one (for example, enter `1` for **Number of Processes** in Fluent Launcher).
- On a given platform, compatibility of data files will be a function of MPI type, but not interconnect.
  - Within the same MPI on a given platform, data files will be compatible regardless of the interconnect (that is, writing with IBM MPI / `ethernet` and reading with IBM MPI / `infiniband` is allowed).
  - Inter-MPI compatibility on the same platform may also exist, since most of the MPIs use the same MPI-O implementation (ROMIO), but this is not guaranteed.
- Parallel data files written on one platform may not necessarily be readable on other platforms. For example, compatibility across 64-bit platforms with the same endianness (byte order) may even exist (for example,

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

lnamd64); however, this is not guaranteed by the standard (since the "native" format is used) and should be investigated on a case-by-case basis.

- No parallel I/O capabilities are currently available for case files.
- The parallel/IO capability is designed to work on true parallel file systems designed for concurrent, single-file access. The capability is not supported on NFS, for example.
- File compression with parallel I/O is supported via asynchronous operation. Inline compression as with standard I/O is not possible. (For details, see Reading and Writing Case and Data Files (p. 45)).
- The general parallel I/O capability is only available with certain MPIs on certain platforms. The supported combinations are listed in Table 3.1: Architecture/MPI Combinations Compatible with Parallel I/O (p. 53). Some of the MPIs currently used on Linux systems do not support MPI-2 features used in the parallel I/O. The default MPIs on all Linux and Windows systems should support the capability.

Important

Not all MPIs on a given platform work with all interconnects. For information about compatibility, see Table 34.7: Supported MPIs for Linux Architectures (Per Interconnect) (p. 2028).

Table 3.1: Architecture/MPI Combinations Compatible with Parallel I/O

Architecture	MPI
lnamd64	ibmmpi, intel, openmpi
win64	ibmmpi, intel, ms

- The parallel I/O capability is available with certain file systems on certain platforms. The supported combinations are listed in Table 3.2: Architecture/File System with Parallel I/O (p. 53).

Table 3.2: Architecture/File System with Parallel I/O

Architecture	File System
lnamd64	Panasas, GPFS, SFS, LUSTRE, PVFS2, MPFS

3.5. Reading Fluent/UNS and RAMPANT Case and Data Files

Case files created by Fluent/UNS 3 or 4 or RAMPANT 2, 3, or 4 can be read into ANSYS Fluent in the same way that current case files are read (see Reading and Writing Case and Data Files (p. 45)). If you read a case file created by Fluent/UNS, ANSYS Fluent selects **Pressure-Based** in the **Solver** group box of the General Task Page (p. 2160). If you read a case file created by RAMPANT, ANSYS Fluent selects **Density-Based** in the **Solver** group box of the General Task Page (p. 2160), as well as **Explicit** from the **Formulation** drop-down menu in the Solution Methods Task Page (p. 2504).

Data files created by Fluent/UNS 4 or RAMPANT 4 can be read into ANSYS Fluent in the same way that current data files are read (see Reading and Writing Case and Data Files (p. 45)).

3.6. Reading and Writing Profile Files

Boundary profiles are used to specify flow conditions on a boundary zone of the solution domain. For example, they can be used to prescribe a velocity field on an inlet plane. For information on boundary

profiles, see Profiles (p. 435). For information about transient profiles, see Defining Transient Cell Zone and Boundary Conditions (p. 448).

For additional information, see the following sections:
3.6.1. Reading Profile Files
3.6.2. Writing Profile Files

3.6.1. Reading Profile Files

To read the boundary profile files, open the **Select File** dialog box by selecting the **File/Read/Profile...** ribbon tab item.

File → **Read** → **Profile...**

This opens the **Select File** dialog box so that you can read a boundary profile with the standard extension `.prof` or a transient profile in tabular format with the standard extension `.ttab`. If a profile in the file has the same name as an existing profile, the old profile will be overwritten.

Note

The maximum number of profiles that can be read into a single Fluent session is 50.

3.6.2. Writing Profile Files

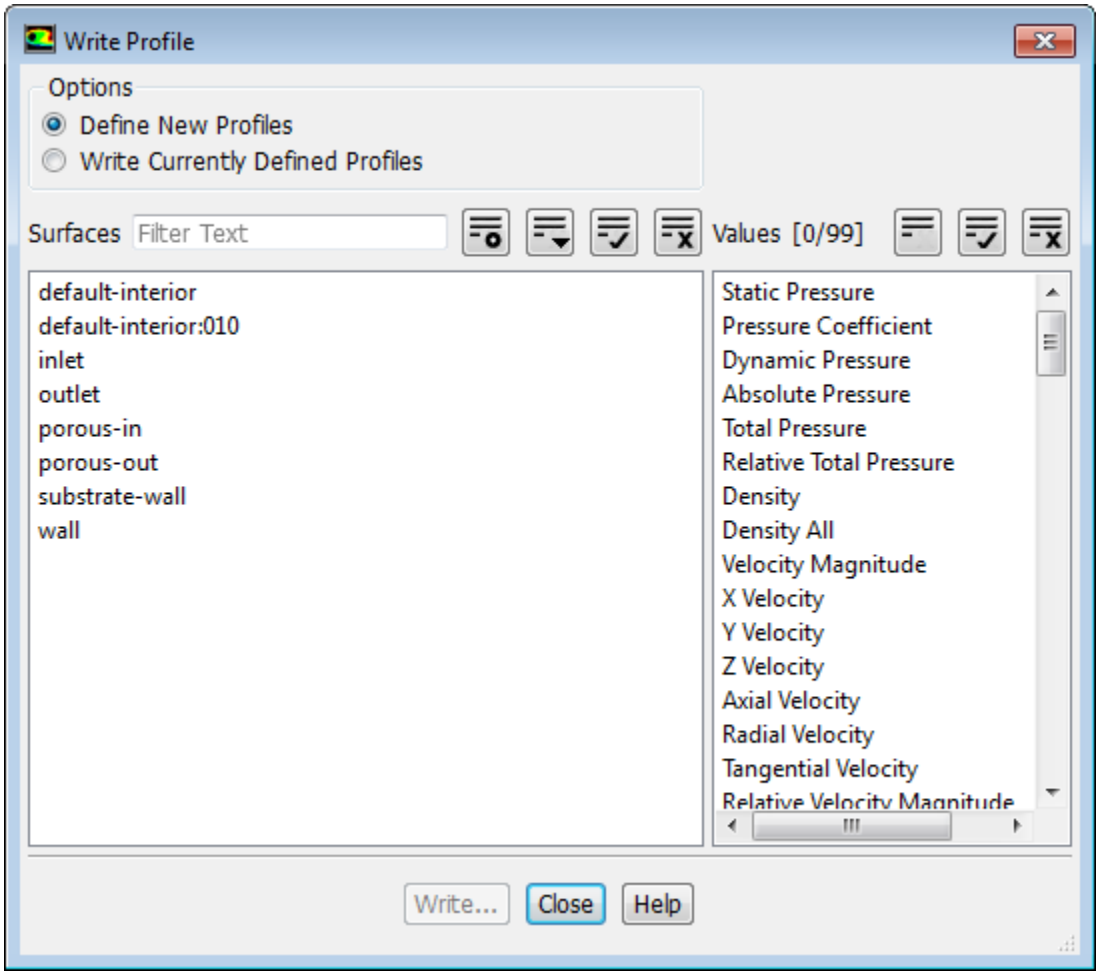
You can also create a profile file from the conditions on a specified boundary or surface. For example, you can create a profile file from the outlet conditions of one case. Then you can read that profile into another case and use the outlet profile data as the inlet conditions for the new case.

To write a profile file, use the Write Profile Dialog Box (p. 2388) (Figure 3.3: The Write Profile Dialog Box (p. 55)).

File → **Write** → **Profile...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.3: The Write Profile Dialog Box



1. Retain the default option of **Define New Profiles**.
2. Select the surface(s) from which you want to extract data for the profile(s) in the **Surfaces** list.
3. Choose the variable(s) for which you want to create profiles in the **Values** list.
4. Click **Write...** and specify the profile file name in the resulting **Select File** dialog box.

   ANSYS Fluent saves the mesh coordinates of the data points for the selected surface(s) and the values of the selected variables at those positions. When you read the profile file back into Fluent, you can select the profile values from the relevant drop-down lists in the boundary condition dialog boxes. The names of the profile values in the drop-down lists will consist of the surface name and the particular variable.
5. Select the **Write Currently Defined Profiles** option:

   - if you have made any modifications to the boundary profiles since you read them in (for example, if you reoriented an existing profile to create a new one).
   - if you want to store all the profiles used in a case file in a separate file.
6. Click **Write...** and specify the file name in the resulting **Select File** dialog box. All currently defined profiles are saved in this file. This file can be read back into Fluent whenever you want to use these profiles again.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.7. Reading and Writing Boundary Conditions

To save all currently defined boundary conditions to a file, enter the `file/write-settings` text command and specify a name for the file.

`file` → `write-settings`

ANSYS Fluent writes the boundary and cell zone conditions, the solver, and model settings to a file using the same format as the "zone" section of the case file. See Case and Data File Formats (p. 2851) for details about the case file format.

To read boundary conditions from a file and to apply them to the corresponding zones in your model, enter the `file/read-settings` text command.

`file` → `read-settings`

ANSYS Fluent sets the boundary and cell zone conditions in the current model by comparing the zone name associated with each set of conditions in the file with the zone names in the model. If the model does not contain a matching zone name for a set of boundary conditions, those conditions are ignored.

If you read boundary conditions into a model that contains a different mesh topology (for example, a cell zone has been removed), check the conditions at boundaries within and adjacent to the region of the topological change. This is important for wall zones.

Note

If the boundary conditions are not checked and some remain uninitialized, the case will not run successfully.

When the `file/read-settings` text command is not used, all boundary conditions get the default settings when a mesh file is imported, allowing the case to run with the default values.

If you want ANSYS Fluent to apply a set of conditions to multiple zones with similar names, or to a single zone with a name you are not sure of in advance, you can edit the boundary-condition file saved with the `file/write-settings` command to include *wildcards* (`*`) within the zone names. For example, if you want to apply a particular set of conditions to **wall-12**, **wall-15**, and **wall-17** in your current model, edit the boundary-condition file so that the zone name associated with the desired conditions is `wall-*`.

Note

The settings file contains only your user-modified settings and does not include default ANSYS Fluent parameters. The default parameters may be updated with each new release. Consequently, the usage of the same settings file in different releases of ANSYS Fluent does not guarantee the same setup, which can cause solution differences.

3.8. Writing a Boundary Mesh

You can write the boundary zones (surface mesh) to a file. This file can be read and used by the meshing mode of Fluent to produce a volume mesh. You may find this feature useful if you are unsatisfied with a mesh obtained from another mesh generation program.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A boundary mesh can be written using the **Select File** dialog box, opened by selecting the **File/Write/Boundary Mesh...** ribbon tab item.

File → Write → Boundary Mesh...

3.9. Reading Scheme Source Files

A Scheme source file can be loaded in three ways: through the menu system as a scheme file, through the menu system as a journal file, or through Scheme itself.

For large source files, use the **Select File** dialog box, opened by selecting the **File/Read/Scheme...** ribbon tab item

File → Read → Scheme...

or use the Scheme `load` function in the console, as shown in the following example:

```
> (load "file.scm")
```

Shorter files can also be loaded with the **File/Read/Journal...** ribbon tab item or the `file/ read-journal` command in the text interface (or its `.` or `source` alias, as shown in the example that follows).

```
>. file.scm

> source file.scm
```

In this case, each character of the file is echoed to the console as it is read, in the same way as if you were typing in the contents of the file.

3.10. Creating and Reading Journal Files

A journal file contains a sequence of ANSYS Fluent commands, arranged as they would be typed interactively into the program or entered through the GUI or TUI. The GUI and TUI commands are recorded as Scheme code lines in journal files. You can also create journal files manually with a text editor. If you want to include comments in your file, be sure to put a semicolon (`;`) at the beginning of each comment line. See Background Execution on Linux Systems in the Getting Started Guide for an example.

The purpose of a journal file is to automate a series of commands instead of entering them repeatedly on the command line. Another use is to produce a record of the input to a program session for later reference, although transcript files are often more useful for this purpose. (For details, see Creating Transcript Files (p. 59)).

Command input is taken from the specified journal file until its end is reached, at which time control is returned to the standard input (usually the keyboard). Each line from the journal file is echoed to the standard output (usually the screen) as it is read and processed.

Important

A journal file is, by design, just a simple record and playback facility. It contains no information about the state in which it was recorded or the state in which it is being played back.

- Be careful not to change the folder while recording a journal file. Also, try to re-create the state in which the journal was written before you read it into the program. For example, if your journal file includes an instruction to save a new file with a specified name, you should check that if a file with that name exists in your folder

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

before you read in your journal file. If a file with that name exists and you read in your journal file, when the program reaches the write instruction, it will prompt for a confirmation to overwrite the old file.

Since the journal file does not contain any response to the confirmation request, ANSYS Fluent cannot continue to follow the instructions of the journal file.

- Other conditions that may affect the program's ability to perform the instructions contained in a journal file can be created by modifications or manipulations that you make within the program.

  For example, if your journal file creates several surfaces and displays data on those surfaces, you must be sure to read in appropriate case and data files before reading the journal file.

Important

At a point of time, only one journal file can be open for recording, but you can write a journal and a transcript file simultaneously. You can also read a journal file at any time.

Whether you choose to type the text command in full or use partial strings (as described in Command Abbreviation (p. 28)), complete commands are recorded in the journal files. Consider the following examples:

- Typing in the TUI

  ```
  solve/set/expert , , yes , ,
  ```

  will be recorded in the journal file as

  ```
  /solve/set/expert yes no yes no no
  ```

  where , or **Enter** signifies default values or entries, as described in Text Prompt System (p. 30).

- Typing in the TUI

  ```
  so set ur mom 0.2 pres 0.4
  ```

  will be recorded in the journal file as two separate commands:

  ```
  /solve/set/under-relaxation/mom 0.2
  /solve/set/under-relaxation/pressure 0.4
  ```

Important

- Only successfully completed commands are recorded. For example, if you stopped an execution of a command using **Ctrl**+**c**, it will not be recorded in the journal file.
- If a GUI event happens while a text command is in progress, the GUI event is recorded first.
- All default values are recorded (as in the first example above).

For additional information, see the following section:
3.10.1. Procedure

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.10.1. Procedure

To start the journaling process, select the **File/Write/Start Journal...** ribbon tab item.

File → Write → Start Journal...

After you enter a name for the file in the **Select File** dialog box, journal recording begins. The **Start Journal...** menu item becomes the **Stop Journal** menu item. You can end journal recording by selecting **Stop Journal**, or by exiting the program.

File → Write → Stop Journal

You can read a journal file into the program using the **Select File** dialog box opened by selecting the **File/Read/Journal...** ribbon tab item.

File → Read → Journal...

Journal files are always loaded in the main (that is, top-level) text menu, regardless of where you are in the text menu hierarchy when you invoke the read command.

3.11. Creating Transcript Files

A transcript file contains a complete record of all standard input to and output from ANSYS Fluent (usually all keyboard and GUI input and all screen output). GUI commands are recorded as Scheme code lines in transcript files. ANSYS Fluent creates a transcript file by recording everything typed as input or entered through the GUI, and everything printed as output in the text window.

The purpose of a transcript file is to produce a record of the program session for later reference. Because they contain messages and other output, transcript files (unlike journal files), cannot be read back into the program.

Important

Only one transcript file can be open for recording at a time, but you can write a transcript and a journal file simultaneously. You can also read a journal file while a transcript recording is in progress.

To start the transcription process, select the **File/Write/Start Transcript...** ribbon tab item.

File → Write → Start Transcript...

After you enter a name for the file in the **Select File** dialog box, transcript recording begins and the **Start Transcript...** menu item becomes the **Stop Transcript** menu item.

You can end transcript recording by selecting **Stop Transcript**, or by exiting the program.

File → Write → Stop Transcript

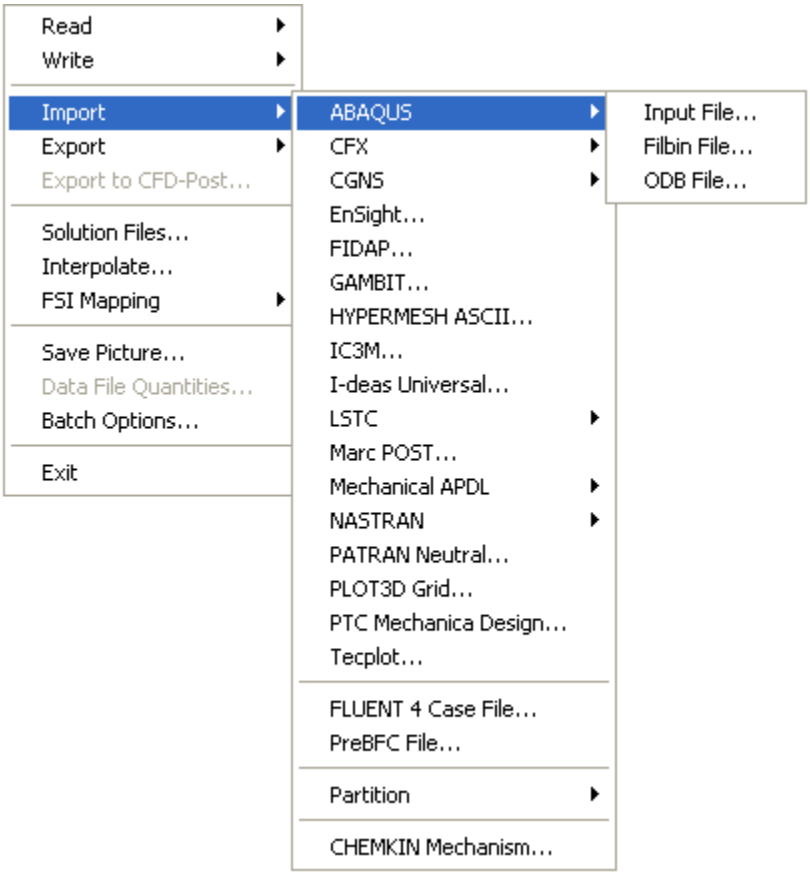
3.12. Importing Files

ANSYS Fluent allows you to import the following file formats:

- ABAQUS `.inp`, `.fil`, and `.odb` files
- CFX `.def` and `.res` files
- CGNS files
- EnSight files
- ANSYS FIDAP files
- GAMBIT files
- HYPERMESH ASCII files
- I-deas Universal files
- LSTC/DYNA keyword input files and state databases
- Marc POST files
- Mechanical APDL `.inp`, `.cdb`, `.rst`, `.rmg`, and `.rfl` files.
- NASTRAN Bulk Data files
- PATRAN Neutral files
- PLOT3D mesh files
- PTC Mechanica Design studies
- Tecplot files

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.4: The Import Menu



Important

Once you have imported any 3rd party file it is recommended that you confirm that the boundary zones and cell zones are set up as you expect.

For information on importing particle history data, see Importing Particle Data (p. 1338).

For additional information, see the following sections:

3.12.1. ABAQUS Files
3.12.2. CFX Files
3.12.3. Meshes and Data in CGNS Format
3.12.4. EnSight Files
3.12.5. ANSYS FIDAP Neutral Files
3.12.6. GAMBIT and GeoMesh Mesh Files
3.12.7. HYPERMESH ASCII Files
3.12.8. I-deas Universal Files
3.12.9. LSTC Files
3.12.10. Marc POST Files
3.12.11. Mechanical APDL Files
3.12.12. NASTRAN Files
3.12.13. PATRAN Neutral Files
3.12.14. PLOT3D Files
3.12.15. PTC Mechanica Design Files
3.12.16. Tecplot Files

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.12.17. Fluent 4 Case Files
3.12.18. PreBFC Files
3.12.19. Partition Files
3.12.20. CHEMKIN Mechanism

3.12.1. ABAQUS Files

To import an ABAQUS input file, use the **File/Import/ABAQUS/Input File...** ribbon tab item.

File → Import → ABAQUS → Input File...

Select this menu item to open the **Select File** dialog box. Specify the name of the ABAQUS **Input File** to be read. The ABAQUS input file (.inp) is a text file that contains the input description of a finite element model for the ABAQUS finite element program. The interface only produces datasets associated with the finite element model, no results of datasets are produced. Element types commonly associated with structural analysis are supported by this file format. There is a list of input keywords that are recognized in the ABAQUS **Input File** [4] (p. 2887).

To import an ABAQUS filbin file, use the **File/Import/ABAQUS/Filbin File...** ribbon tab item.

File → Import → ABAQUS → Filbin File...

Select this menu item to open the **Select File** dialog box. Specify the name of the ABAQUS **Filbin File** to be read. This output file has a `.fil` extension and consists of finite element model and results data.

To import an ABAQUS ODB file, use the **File/Import/ABAQUS/ODB File...** ribbon tab item.

File → Import → ABAQUS → ODB File...

Select this menu item to open the **Select File** dialog box. Specify the name of the ABAQUS **ODB File** to be read. This output database file has a `.odb` extension and consists of finite element model and results data in the OpenDocument format.

3.12.2. CFX Files

To import a CFX definition file, use the **File/Import/CFX/Definition File...** ribbon tab item.

File → Import → CFX → Definition File...

Select this menu item to open the **Select File** dialog box. Specify the name of the CFX **Definition File** to be read. Fluent reads mesh information from the CFX file with `.def` extensions. For information about importing CFX files, see CFX Files (p. 141).

To import a CFX result file, use the **File/Import/CFX/Result File...** ribbon tab item.

File → Import → CFX → Result File...

In the **Select File** dialog box, specify the name of the CFX **Result File** to be read. Those imported files will have `.res` extensions.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that the **Create Zones from CCL Physics Data** option in the **Select File** dialog boxes allows you to create zones from the physics data objects or the primitive mesh region objects.

Important

CFX file import is available for 3D cases only.

3.12.3. Meshes and Data in CGNS Format

To import meshes in CFD general notation system (CGNS) format (`.cgns`) into ANSYS Fluent, use the **File/Import/CGNS/Mesh...** ribbon tab item.

File → Import → CGNS → Mesh...

To import a mesh and the corresponding CGNS data, use the **File/Import/CGNS/Mesh & Data...** ribbon tab item.

File → Import → CGNS → Mesh & Data...

To import only the CGNS data, use the **File/Import/CGNS/Data...** ribbon tab item.

File → Import → CGNS → Data...

Table 3.3: CGNS Variables Supported by ANSYS Fluent

CGNS Variable Name	ANSYS Fluent Name
Pressure	pressure
ReynoldsStressXX	uu-stress
ReynoldsStressXY	uv-stress
ReynoldsStressXZ	uw-stress
ReynoldsStressYY	vv-stress
ReynoldsStressYZ	vw-stress
ReynoldsStressZZ	ww-stress
Temperature	temperature
TurbulentDissipation	epsilon
TurbulentDissipationRate	omega
TurbulentEnergyKinetic	k
VelocityX	x-velocity
VelocityY	y-velocity
VelocityZ	z-velocity

Important

- To import data correctly, first import the mesh using the mesh only option (**Mesh...**) and set up the boundary conditions. For example, if a boundary zone is of type **pressure-outlet** and is read as **outlet**, it should be changed to **pressure-outlet** before importing

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the data. Then select the appropriate model and read the data using the data only option (**Data...**)

- The new and original meshes should have the same zones, numbered in the same order. A warning is issued if they do not, because inconsistencies can create problems with the boundary conditions.
- ANSYS Fluent defaults to the **Laminar Viscous Model** after reading a CGNS mesh. You must select the appropriate model before reading data since ANSYS Fluent selectively imports only those values from the CGNS file required for a given model. For example, if **Epsilon** is available in the CGNS file, the model must be changed to **k-epsilon** to ensure that ANSYS Fluent will import this quantity.

3.12.4. EnSight Files

You can import an EnSight file using the **File/Import/EnSight...** ribbon tab item.

File → Import → EnSight...

This file format is applied to both unstructured and structured data, where each part contains its own local coordinate array. The EnSight Gold software package, which uses this file format, allows you to analyze, visualize, and communicate engineering datasets. It allows you to take full advantage of parallel processing and rendering and supports a range of virtual reality devices. Furthermore, it enables real-time collaboration.

When selecting this option, the **Select File** dialog box will appear, where you will specify a file name. This file will have an `.encas` or `.case` extension.

Only the mesh file is read into ANSYS Fluent, and any data present is discarded.

3.12.5. ANSYS FIDAP Neutral Files

You can read an ANSYS FIDAP neutral file using the **File/Import/FIDAP...** ribbon tab item.

File → Import → FIDAP...

In the **Select File** dialog box, specify the name of the ANSYS FIDAP **Neutral File** to be read. This file will have an `.FDNEUT` or `.unv` file extension. ANSYS Fluent reads mesh information and zone types from the ANSYS FIDAP file. You must specify boundary conditions and other information after reading this file. For information about importing ANSYS FIDAP Neutral files, see ANSYS FIDAP Neutral Files (p. 145).

3.12.6. GAMBIT and GeoMesh Mesh Files

If you have saved a neutral file from GAMBIT, rather than an ANSYS Fluent mesh file, you can import it into ANSYS Fluent using the **File/Import/GAMBIT...** ribbon tab item.

File → Import → GAMBIT...

For information about importing files from GAMBIT and GeoMesh, see GAMBIT Mesh Files (p. 136) and GeoMesh Mesh Files (p. 136).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.12.7. HYPERMESH ASCII Files

You can read a HYPERMESH ASCII file using the **File/Import/HYPERMESH ASCII...** ribbon tab item.

File → Import → HYPERMESH ASCII...

HYPERMESH is a high-performance finite element pre- and postprocessor for popular finite element solvers, allowing engineers to analyze product design performance in a highly interactive and visual environment.

When selecting this option, the **Select File** dialog box will appear, where you will specify a file name. This file should have an `.hm`, `.hma`, or `.hmascii` extension.

3.12.8. I-deas Universal Files

I-deas Universal files can be read into ANSYS Fluent with the**File/Import/I-deas Universal...** ribbon tab item.

File → Import → I-deas Universal...

Select the **I-deas Universal...** menu item to open the **Select File** dialog box.

Specify the name of the I-deas Universal file to be read. Fluent reads mesh information and zone types from the I-deas Universal file. For information about importing I-deas Universal files, seeI-deas Universal Files (p. 137).

3.12.9. LSTC Files

To import an LSTC input file, use the **File/Import/LSTC/Input File...** ribbon tab item.

File → Import → LSTC → Input File...

The LSTC input file is a text file that contains the input description of a finite element model for the LS-DYNA finite element program. This interface only produces datasets associated with the mesh, no results datasets are produced. The element types commonly associated with structural analysis are supported.

LSTC input files have the following file extensions: `.k`, `.key`, and `.dyn`

To import an LSTC state file, use the **File/Import/LSTC/State File...** ribbon tab item.

File → Import → LSTC → State File...

The state file consists of three major sections: control data, geometry data, and state data. Each dataset in the state data section corresponds to the time and global data items associated with each state on the database. Dataset attributes include such things as time, energy, and momentum.

An LSTC state file has a `.d3plot` file extension.

3.12.10. Marc POST Files

Marc POST files can be read into ANSYS Fluent using the **File/Import/Marc POST...** ribbon tab item.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File → Import → Marc POST...

Select the **Marc POST...** menu item and in the **Select File** dialog box, specify the name of the file to be read.

These files are generated using MSC Marc, a nonlinear finite element program. MSC Marc allows you to study deformations that exceed the linear elastic range of some materials, enabling you to assess the structural integrity and performance of the material. It also allows you to simulate deformations that are part-to-part or part-to-self contact under a range of conditions.

3.12.11. Mechanical APDL Files

To import a Mechanical APDL input file, use the **File/Import/Mechanical APDL/Input File...** ribbon tab item.

File → Import → Mechanical APDL → Input File...

Select this menu item to open the **Select File** dialog box. Specify the name of the Mechanical APDL **Prep7 File** to be read. Fluent reads mesh information from the Mechanical APDL file with `.ans`, `.neu`, `.cdb`, and `.prep7` extensions. For information about importing Mechanical APDL files, see Mechanical APDL Files (p. 140).

To import a Mechanical APDL result file, use the **File/Import/Mechanical APDL/Result File...** ribbon tab item.

File → Import → Mechanical APDL → Result File...

In the **Select File** dialog box. Specify the name of the Mechanical APDL **Result File** to be read. Those imported files will have `.rfl`, `.rst`, `.rth`, and `.rmg` extensions.

3.12.12. NASTRAN Files

You can read NASTRAN Bulkdata files into ANSYS Fluent with the **File/Import/NASTRAN/Bulkdata File...** ribbon tab item.

File → Import → NASTRAN → Bulkdata File...

When you select the **Bulkdata File...** menu item, the **Select File** dialog box will appear and you will specify the name of the NASTRAN **File** to be read. This file will have `.nas`, `.dat`, `.bdf` file extensions. Fluent reads mesh information from the NASTRAN file. For information about importing NASTRAN files, see NASTRAN Files (p. 138).

To import NASTRAN Op2 files into ANSYS Fluent, use the **File/Import/NASTRAN/Op2 File...** ribbon tab item.

File → Import → NASTRAN → Op2 File...

In the **Select File** dialog box, specify the name of the NASTRAN **Output2 File** to be read. This file is an output binary data file that contains data used in the NASTRAN finite element program. This file will have `.op2` file extension.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.12.13. PATRAN Neutral Files

To read a PATRAN Neutral file zoned by named components (that is, a file in which you have grouped nodes with the same specified group name), use the **File/Import/PATRAN Neutral...** ribbon tab item.

File → Import → PATRAN Neutral...

Selecting this menu item opens the **Select File** dialog box. Specify the name of the PATRAN Neutral file to be read (extension `.neu`, `.out`, or `.pat`). Fluent reads mesh information from the PATRAN Neutral file. For information about importing PATRAN Neutral files, see PATRAN Neutral Files (p. 139).

3.12.14. PLOT3D Files

To import a PLOT3D mesh file, use the **File/Import/PLOT3D Grid...** ribbon tab item.

File → Import → PLOT3D Grid...

The PLOT3D mesh files have `.p3d`, `.bin`, `.x`, `.xyz`, or `.grd` file extensions.

These file formats may be formatted, unformatted or binary.

3.12.15. PTC Mechanica Design Files

To import a PTC Mechanica Design file, use the **File/Import/PTC Mechanica Design...** ribbon tab item.

File → Import → PTC Mechanica Design...

This will open the **Select File** dialog box. Specify the name of the neutral file to be read.

The PTC Mechanica Design file contains analysis, model and results data. Only the binary form of the results data files is supported.

The form of the file must have the `.neu` extension.

Important

Mechanica results consists of an entire directory structure of files, called a "study" in Mechanica terminology, which must be used in exactly the form that Mechanica originally generates it. ANSYS Fluent's VKI interface keys on the `.neu` file and can traverse the directory structure from there to access the other files that it needs.

3.12.16. Tecplot Files

To import a Tecplot file, use the **File/Import/Tecplot...** ribbon tab item.

File → Import → Tecplot...

This will open the **Select File** dialog box. Specify the name of the neutral file to be read.

The Tecplot file is a binary file. Only the mesh is read into ANSYS Fluent and any data present is discarded.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The form of the file must have the `.plt` extension. ANSYS Fluent supports the importation of polyhedral cells and files created by Tecplot version 7.1–11.2, except for version 11.0 (which is unsupported).

3.12.17. Fluent 4 Case Files

You can read a Fluent 4 case file using the **File/Import/Fluent 4 Case File...** ribbon tab item.

File → Import → Fluent 4 Case File...

Select the **Fluent 4 Case File...** menu item to open the **Select File** dialog box. Specify the name of the Fluent 4 case file to be read. ANSYS Fluent reads *only* mesh information and zone types from the Fluent 4 case file. You must specify boundary and cell zone conditions, model parameters, material properties, and other information after reading this file. For information about importing Fluent 4 case files, see FLUENT 4 Case Files (p. 144).

3.12.18. PreBFC Files

You can read a PreBFC structured mesh file into ANSYS Fluent using the **File/Import/PreBFC File...** ribbon tab item.

File → Import → PreBFC File...

Select the **PreBFC File...** menu item to open the **Select File** dialog box. Specify the name of the PreBFC structured mesh file to be read. Fluent reads mesh information and zone types from the PreBFC mesh file. For information about importing PreBFC mesh files, see PreBFC Mesh Files (p. 136).

3.12.19. Partition Files

To perform METIS partitioning on an unpartitioned mesh, use the **File/Import/Partition/Metis...** ribbon tab item.

File → Import → Partition → Metis...

You may also partition each cell zone individually, using the **File/Import/Partition/Metis Zone...** ribbon tab item.

File → Import → Partition → Metis Zone...

See Using the Partition Filter (p. 2052) for detailed information about partitioning.

3.12.20. CHEMKIN Mechanism

To import a CHEMKIN format, you can import the mechanism file into ANSYS Fluent using the **File/Import/CHEMKIN Mechanism...** ribbon tab item (Figure 14.5: The CHEMKIN Mechanism Import Dialog Box for Volumetric Kinetics (p. 969)).

File → Import → CHEMKIN Mechanism...

See Importing a Volumetric Kinetic Mechanism in CHEMKIN Format (p. 967) for detailed information on importing a CHEMKIN Mechanism file.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.13. Exporting Solution Data

The current release of ANSYS Fluent allows you to export data to ABAQUS, Mechanical APDL, Mechanical APDL Input, ASCII, AVS, ANSYS CFD-Post, CGNS, Data Explorer, EnSight, FAST, FIELDVIEW, I-deas, NASTRAN, PATRAN, RadTherm, and Tecplot formats. Exporting Solution Data after a Calculation (p. 70) explains how to export solution data in these formats after the calculation is complete, and ABAQUS Files (p. 71) to Tecplot Files (p. 83) provide specific information for each type of **File Type**. For information about exporting solution data during transient flow solutions, see Exporting Data During a Transient Calculation (p. 86).

For NASTRAN, ABAQUS, Mechanical APDL Input, I-deas Universal, and PATRAN file formats, the following quantities are exported [4] (p. 2887):

- Nodes, Elements
- Node Sets (Boundary Conditions)
- Temperature
- Pressure
- Heat Flux
- Heat Transfer Coefficient
- Force

  To generate the force data that is exported for nodes at boundaries, ANSYS Fluent performs the following steps:

  1. Facial force for each wall face is calculated by summing the pressure force, viscous force and surface tension force of the face.
  2. Partial force for each wall face is calculated by dividing its facial force by its number of shared nodes.
  3. Total force for each wall node is calculated by summing the partial forces of all the wall faces sharing that node.

For additional information, see the following section:
3.13.1. Exporting Limitations

3.13.1. Exporting Limitations

Note the following limitations when exporting solution data:

- When using the parallel version of ANSYS Fluent, you can only export to the following packages:
  - ABAQUS
  - ANSYS CFD-Post
  - ASCII
  - CGNS

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - EnSight Case Gold
  - Fieldview Unstructured
  - I-deas Universal
  - Mechanical APDL Input
  - NASTRAN
  - PATRAN
  - RadTherm

    Note that parallel RadTherm export is only available for surface data.

  - Tecplot

  The exported file will be written by the host process (see Introduction to Parallel Processing (p. 2009)). Note that the memory required to write the file may exceed the memory available to the host process.

- When using the parallel version of ANSYS Fluent, the only packages to which you can export custom field functions are the following:

  - ANSYS CFD-Post
  - EnSight Case Gold
  - Fieldview Unstructured

  For further information about custom field functions, see Custom Field Functions (p. 2003).

- The solution mode of Fluent cannot import surfaces. Consequently, if you export a file from ANSYS Fluent with surfaces selected, you may not be able to read these files back into the solution mode. However, the meshing mode of Fluent can import surface data (see the Fluent Meshing User's Guide for details).

- ANSYS Fluent supports exporting polyhedral data only for ASCII, ANSYS CFD-Post, CGNS, EnSight Case Gold, NASTRAN, RadTherm, Tecplot and Fieldview Unstructured file formats. For further details, see ASCII Files (p. 74), ANSYS CFD-Post-Compatible Files (p. 74), CGNS Files (p. 75), EnSight Case Gold Files (p. 76), NASTRAN Files (p. 82), RadTherm Files (p. 83) and FieldView Unstructured Files (p. 80).

- If the files that are exported during multiple transient simulations are to be used as a set, you must make sure that all of the simulations are run on the same platform, using the same number of processors. This ensures that all of the files are compatible with each other.

- Data on user defined/created surfaces is not available for export to EnSight Case Gold file format.

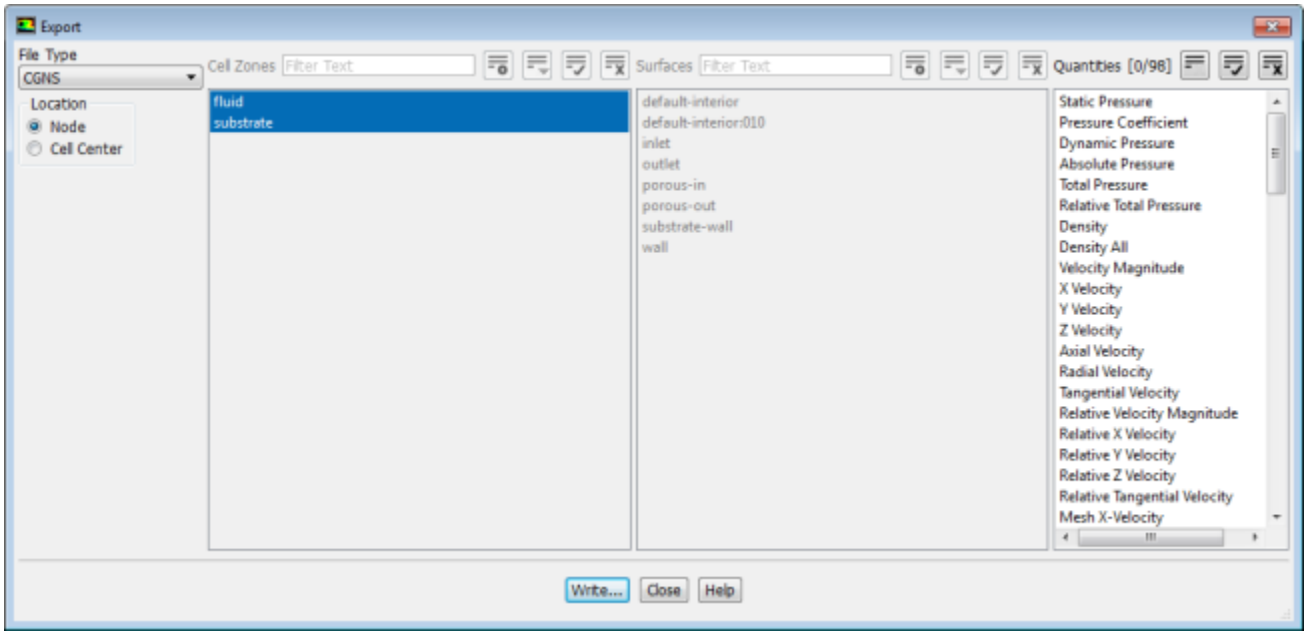
3.14. Exporting Solution Data after a Calculation

To export solution data to a different file format after a calculation is complete, use the Export Dialog Box (p. 2644) (Figure 3.5: The Export Dialog Box (p. 71)).

File → Export → Solution Data...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.5: The Export Dialog Box



Information concerning the necessary steps and available options for each **File Type** are listed in ABAQUS Files (p. 71) to Tecplot Files (p. 83).

For details about general limitations for exporting solution data and the manner in which it is exported, see Exporting Solution Data (p. 69).

For additional information, see the following sections:

3.14.1. ABAQUS Files
3.14.2. Mechanical APDL Files
3.14.3. Mechanical APDL Input Files
3.14.4. ASCII Files
3.14.5. AVS Files
3.14.6. ANSYS CFD-Post-Compatible Files
3.14.7. CGNS Files
3.14.8. Data Explorer Files
3.14.9. EnSight Case Gold Files
3.14.10. FAST Files
3.14.11. FAST Solution Files
3.14.12. FieldView Unstructured Files
3.14.13. I-deas Universal Files
3.14.14. NASTRAN Files
3.14.15. PATRAN Files
3.14.16. RadTherm Files
3.14.17. Tecplot Files

3.14.1. ABAQUS Files

Select ABAQUS from the **File Type** drop-down list and choose the surface(s) for which you want to write data in the **Surfaces** list. If no surfaces are selected, the entire domain is exported.

When the **Energy Equation** is enabled in the Energy Dialog Box (p. 2175), you can choose the loads to be written based on the kind of finite element analysis you intend to undertake. By selecting **Structural**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in the **Analysis** list, you can select the following **Structural Loads**: **Force**, **Pressure**, and **Temperature**. By selecting **Thermal** in the **Analysis** list, you can select the following **Thermal Loads**: **Temperature**, **Heat Flux**, and **Heat Trans Coeff**. Note the following limitations with these loads:

- When the **Energy Equation** is disabled, only the **Structural Loads** options of **Force** and **Pressure** are available.
- Loads are written only on boundary walls when the entire domain is exported (that is, if no **Surfaces** are selected).

Click the **Write...** button to save the file, using the **Select File** dialog box. The exported file format of ABAQUS (`file.inp`) contains coordinates, connectivity, zone groups, and optional loads.

Export of data to ABAQUS is valid only for solid zones or for those surfaces that lie at the intersection of solid zones. Temperature data is exported for the whole domain.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See Exporting Data During a Transient Calculation (p. 86) for the complete details.

3.14.2. Mechanical APDL Files

Export to Mechanical APDL can only be opened using the `file/export/mechanical-apdl` text command. You will be prompted to enter a `Mechanical APDL file name` and the `Zone to Export`.

The file written is a Mechanical APDL results file with a `.rfl` extension. This file preserves the cell zones defined in ANSYS Fluent. The Mechanical APDL file is a single file containing coordinates, connectivity, and the scalars listed below:

```
''x-velocity'', ''y-velocity'', ''z-velocity'', ''pressure'',
''temperature'', ''turb-kinetic-energy'', ''turb-diss-rate'',
''density'', ''viscosity-turb'', ''viscosity-lam'', ''viscosity-eff'',
''thermal-conductivity-lam'', ''thermal-conductivity-eff'',
''total-pressure'', ''total-temperature'', ''pressure-coefficient'',
''mach-number'', ''stream-function'', ''heat-flux'',
''heat-transfer-coef'', ''wall-shear'', ''specific-heat-cp''
```

Important

Export to ANSYS is available on a limited number of platforms.

To read this file into Mechanical APDL, do the following:

1. In Mechanical APDL, expand the **General Postproc** item in the **ANSYS Main Menu**, and click **Data & File Opts**. Then perform the following steps in the **Data and File Options** dialog box that opens.
   a. Make sure that **Read single result file** is selected.
   b. Click the **Browse...** button to open the **Open** dialog box, and select the `.rfl` file generated by ANSYS Fluent.
   c. Click **Open** in the **Open** dialog box.
   d. Click **OK** in the **Data and File Options** dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. In the **ANSYS Main Menu**, click **Results Summary** in the expanded **General Postproc** list. A **SET,LIST Command** window will open, displaying a summary of the file contents.

3. In the expanded **General Postproc** list, expand the **Read Results** item and click **First Set** to read the data from the file.

4. Display the data from the file. You can display mesh information by selecting either the **Plot/Nodes** menu item or the **Plot/Elements** menu item from the Mechanical APDL menu bar. You can display results by selecting either the **Plot/Results/Contour Plot/Nodal Solution...** menu item or the **Plot/Results/Contour Plot/Elem Solution...** menu item, and then using the dialog box that opens.

You have the option of using the **Execute Commands** dialog box to export data to Mechanical APDL at specified intervals during the calculation. The text command for exporting can be entered directly into the **Command** text box (or the ANSYS Fluent console, if you are defining a macro). It is of the following form:

`file/export/mechanical-apdl` *file_name list_of_cell_zones* `()`

where

- *file_name* specifies the name (without the extension) of the file that you want to write. You should include a special character string in the file name, so that Fluent assigns a new name to each file it saves. For details, see Automatic Numbering of Files (p. 43). By using a special character string, you can have the files numbered by time step, flow time, and so on.

- *list_of_cell_zones* specifies the list of cell zones (separated by commas) from which you want to export data. The `()` input terminates the list. For example, the input `fluid-rotor, fluid-stator()` will select the cell zones named **fluid-rotor** and **fluid-stator**.

See Executing Commands During the Calculation (p. 1638) for information about executing commands and creating and using command macros.

3.14.3. Mechanical APDL Input Files

Select Mechanical APDL Input from the **File Type** drop-down list and choose the surface(s) for which you want to write data in the **Surfaces** list. If no surfaces are selected, the entire domain is exported.

When the **Energy Equation** is enabled in the Energy Dialog Box (p. 2175), you can choose the loads to be written based on the kind of finite element analysis you intend to undertake. By selecting **Structural** in the **Analysis** list, you can select the following **Structural Loads**: **Force**, **Pressure**, and **Temperature**. By selecting **Thermal** in the **Analysis** list, you can select the following **Thermal Loads**: **Temperature**, **Heat Flux**, and **Heat Trans Coeff**. Note the following limitations with these loads:

- When the **Energy Equation** is disabled, only the **Structural Loads** options of **Force** and **Pressure** are available.

- Loads are written only on boundary walls when the entire domain is exported (that is, if no **Surfaces** are selected).

Click the **Write...** button to save the file, using the **Select File** dialog box. ANSYS Fluent exports an input file that contains Mechanical APDL finite element information including nodes, elements, and loads that can be used to do finite element analysis in Mechanical APDL with minimal effort. The file format is written in `.cdb` format. The export of Mechanical APDL Input files is in ASCII format and therefore is available on all platforms.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See Exporting Data During a Transient Calculation (p. 86) for the complete details.

3.14.4. ASCII Files

Select **ASCII** from the **File Type** drop-down list and choose the surface(s) for which you want to write data in the **Surfaces** list. If no surfaces are selected, the entire domain is exported. Also select the variable(s) for which data is to be saved in the **Quantities** list.

When exporting ASCII files, you have the following options:

- Select the **Location** from which the values of scalar functions are to be taken. If you specify the data **Location** as **Node**, then the data values at the node points are exported. If you choose **Cell Center**, then the data values from the cell centers are exported. For boundary faces, it is the face center values that are exported when the **Cell Center** option is selected.
- Select the **Delimiter** separating the fields (**Comma** or **Space**).

Click the **Write...** button to save the file, using the **Select File** dialog box. ANSYS Fluent will export a single ASCII file containing coordinates, optional loads, and specified scalar function data.

Important

ANSYS Fluent supports exporting polyhedral data to ASCII.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See Exporting Data During a Transient Calculation (p. 86) for the complete details.

3.14.5. AVS Files

Select **AVS** from the **File Type** drop-down list and specify the scalars you want in the **Quantities** list.

Click the **Write...** button to save the file, using the **Select File** dialog box. An AVS version 4 UCD file contains coordinate and connectivity information and specified scalar function data.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See Exporting Data During a Transient Calculation (p. 86) for the complete details.

3.14.6. ANSYS CFD-Post-Compatible Files

To export data to files that are compatible with ANSYS CFD-Post, select **CFD-Post Compatible** from the **File Type** drop-down list. Next, specify the **Cell Zones** that contain the data you want exported; note that by default, all of the cell zones will be exported. If you want to export solution data for particular surfaces, then select them from the **Surfaces** list. Once you have chosen your cells zones and/or surfaces you must select the **Quantities** that you want exported. When selecting the variables, be sure to include any variable that was used to create an isosurface or clipped surface (as described in Isosurfaces (p. 1728) and Clipping Surfaces (p. 1730), respectively); otherwise, the surface will not be created properly if you try to read the exported state file in CFD-Post.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specify the format of the `.cdat` file by selecting either **Binary** or **ASCII** from the **Format** list. The advantage of the binary format is that it takes less time to load the exported data into ANSYS CFD-Post and requires less storage space. Note that the format for the `.cst` will always be ASCII.

By default, Fluent will write a case file (that is, `.cas`) along with the **CFD-Post Compatible** files. If you do not want such a case file for any reason (for example., to improve I/O performance or save disc space), disable the **Write Case File** option. Disabling this option is only recommended if you are performing multiple export operations for a case file that is not changing.

Click the **Write...** button to save the files, using the **Select File** dialog box. A `.cdat` file is written, containing the specified variable data for the specified cell zones and all of the boundary zones. A state file (that is, `.cst`) is also written, which contains the following surfaces that you created in Fluent for postprocessing: point surfaces, line surfaces, plane surfaces, isosurfaces, and clipped surfaces (see Creating Surfaces and Cell Registers for Displaying and Reporting Data (p. 1709)).

Note

Imprinted surfaces defined in ANSYS Fluent are not available for postprocessing in ANSYS CFD-Post.

Important

When you read the `.cdat` file into ANSYS CFD-Post, the application will attempt to read in the case file that produced the data by looking in the folder for a `.cas` file with the same prefix. If the case file is not in that folder (e.g., if you disabled the **Write Case File** option when exporting), ANSYS CFD-Post will prompt you to specify the appropriate case file.

Important

Before loading the `.cst` file into ANSYS CFD-Post, make sure that the `.cas` and `.cdat` files are already loaded for the session.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See Exporting Data During a Transient Calculation (p. 86) for the complete details.

3.14.7. CGNS Files

Select **CGNS** from the **File Type** drop-down list and specify the scalars you want in the **Quantities** list.

Select the **Location** from which the values of scalar functions are to be taken. If you specify the data **Location** as **Node**, then the data values at the node points are exported. If you choose **Cell Center**, then the data values from the cell centers are exported. For boundary faces, it is the face center values that are exported when the **Cell Center** option is selected.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Click the **Write...** button to save the file, using the **Select File** dialog box. CGNS (CFD general notation system) is a single file (for example, `file.cgns`) containing coordinates, connectivity, zone information, velocity, and selected scalars.

Important

ANSYS Fluent supports exporting polyhedral data to CGNS.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See Exporting Data During a Transient Calculation (p. 86) for the complete details.

3.14.8. Data Explorer Files

Select **Data Explorer** from the **File Type** drop-down list and choose the surface(s) for which you want to write data in the **Surfaces** list. If no surfaces are selected, the entire domain is exported. Also specify the scalars you want in the **Quantities** list.

Important

When you are exporting data for Data Explorer, EnSight Case Gold, or I-deas Universal and the reference zone is not a stationary zone, the data in the velocity fields is exported by default as velocities relative to the motion specification of that zone. This data is always exported, even if you do not choose to export any scalars. Any velocities that you select to export as scalars in the **Quantities** list (for example, **X Velocity**, **Y Velocity**, **Radial Velocity**, and so on) are exported as absolute velocities. For all other types of exported files, the velocities exported by default are absolute velocities.

Click the **Write...** button to save the file, using the **Select File** dialog box. A single file (for example, `file.dx`) is exported, containing coordinate, connectivity, velocity, and specified function data.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See Exporting Data During a Transient Calculation (p. 86) for the complete details.

3.14.9. EnSight Case Gold Files

When exporting to EnSight, you can choose to export the data associated with a single data file, or you can export the data associated with multiple files that were generated by a transient solution.

- To export the solution data from a single data file, read in the case and data file and open the **Export** dialog box.

  File → **Export** → **Solution Data...**

  Then perform the following steps in the **Export** dialog box:

  1. Select **EnSight Case Gold** from the **File Type** drop-down list.

  2. Select the **Location** from which the values of scalar functions are to be taken. If you specify the data **Location** as **Node**, then the data values at the node points are exported. If you choose **Cell Center**, then

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the data values from the cell centers are exported. For boundary faces, it is the face center values that are exported when the **Cell Center** option is selected.

3. Specify the format of the file by selecting either **Binary** or **ASCII** from the **Format** list. The advantage of the binary format is that it takes less time to load the exported files into EnSight.

4. (optional) Select the **Cell Zones** from which you want data exported (you must select at least one zone). By default, all of the cell zones are selected.

5. (optional) Select the **Interior Zone Surfaces** from which you want data exported. By default, the data being exported is taken from the entire ANSYS Fluent domain. The **Interior Zone Surfaces** selection list allows you to also specify that the data be taken from selected zone surfaces whose **Type** is identified as **interior** in the **Boundary Conditions** task page.

   Note

   Data on user defined/created surfaces is not available for export to EnSight Case Gold file format.

6. Select the scalars you want to write from the **Quantities** selection list.

7. Click the **Write...** button to open the **Select File** dialog box, which you can use to save a file that contains the specified variable data for the specified cell zones, the specified interior zone surfaces, and all of the boundary zones.

- To export solution data associated with multiple files that were generated by a transient solution, you must first make sure that the following criteria are met:

  – All of the relevant case and data files must be in the working folder.

  – The data files must be separated by a consistent number of time steps.

  Next, enter the following text command in the console:

  `file → transient-export → ensight-gold-from-existing-files`

  Then, enter responses to the following prompts in the console:

  1. `EnSight Case Gold file name`

     Enter the name you want assigned to the exported files.

  2. `Case / Data file selection by base name?`

     Enter `yes` if the case and data files share a “base name” (that is, a common initial string of characters). The alphanumeric order of the full names must correspond to the order in which the files were created. You will then be prompted to enter the base name at the `Case / Data file base name` prompt that follows. For example, with a set of files named `elbow-0001`, `elbow-0002`, `elbow-0003`, and so on, enter `elbow-` for the base name.

     Enter `no` if you have created an ASCII file in the working folder that lists the names of the data files in order of when they were created. The file should list one data file name per line. You will then be prompted to enter the name of this file at the `Provide the file name which`

`contains the data file names` prompt that follows. Note that you must include the file extension if any in your entry at the prompt.

3. `Specify Skip Value`

   Enter an integer value to specify the number of files you want to skip in between exporting files from the sequence. For example, enter `1` to export every other file, enter `2` to export every third file, and so on.

4. `Cell-Centered?`

   Enter `yes` if you want to export the data values from the cell centers (or face center values, for boundary faces).

   Enter `no` if you want to export the data values from the node points.

5. `Write separate file for each time step for each variable?`

   Enter `yes` if you want separate EnSight Case Gold files written for each time step. Otherwise, all of the data for the `.scl1` and `.vel` files will be combined into a single file for each.

6. `Write in binary format?`

   Enter `yes` to write the files in binary format. Otherwise, they will be written in ASCII format. The advantage of the binary format is that it takes less time to load the exported files into EnSight.

7. `Specify Data File Frequency`

   Enter the number of time steps between the data files being exported.

8. `Separate case file for each time step?`

   Enter `no` if all the data files were generated from the same case file (that is, the simulation involved a static mesh). Note that the name of the case file must be the same (not including the extension) as the name of the first data file in the sequence.

   Enter `yes` if the data files were generated from the different case files (that is, the simulation involved a sliding or dynamic mesh). Note that the names of the case files must be the same (not including the extension) as the names of the corresponding data files.

9. `Read the case file?`

   Enter `no` if the first (or only, for a static mesh) case file is already in memory.

   Enter `yes` if the first (or only, for a static mesh) case file is *not* already in memory.

10. `cell zone id/name(1)`

    Enter the name or ID of any cell zone from which you want data exported. By default, the data being exported is taken from the entire ANSYS Fluent domain. After you specify the first cell zone, you will be prompted to specify the second one, and so on, until you press **Enter** without typing any characters.

11. `Interior Zone Surfaces(1)`

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Enter the name of any interior zone surface from which you want data exported. By default, the data being exported is taken from the entire ANSYS Fluent domain. This prompt allows you to also specify that the data be taken from selected zone surfaces whose **Type** is identified as **interior** in the **Boundary Conditions** task page. After you specify the first interior zone surface, you will be prompted to specify the second one, and so on, until you press **Enter** without typing any characters.

12. `EnSight Case Gold scalar(1) else q to continue`

    Enter in the first scalar quantity you want exported. You can press the **Enter** key to print a list of available scalar quantities in the console. After you enter the first quantity, you will be prompted to enter the second quantity, and so on, until you enter `q`. The EnSight Case Gold files will then be written.

When exporting to EnSight Case Gold, files will be created with the following four formats:

- A geometry file (for example, `file.geo`) containing the coordinates and connectivity information.
- A velocity file (for example, `file.vel`) containing the velocity.
- A scalar file (for example, `file.scl1`) for each selected variable or function.
- An EnSight case file (for example, `file.encas`) that contains details about the other exported files.

Important

- For non-stationary reference zones, all the velocities are exported to EnSight as velocities relative to the selected reference zone. See the informational note in Data Explorer Files (p. 76) for further details.
- ANSYS Fluent supports exporting polyhedral data to EnSight.

You also have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. For details, see Exporting Data During a Transient Calculation (p. 86).

3.14.10. FAST Files

This file type is valid only for triangular and tetrahedral meshes. Select **FAST** from the **File Type** drop-down list and select the scalars you want to write in the **Quantities** list.

Click the **Write...** button to save the file for the specified function(s), using the **Select File** dialog box. The following files are written:

- A mesh file in extended Plot3D format containing coordinates and connectivity.
- A velocity file containing the velocity.
- A scalar file for each selected variable or function.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See Exporting Data During a Transient Calculation (p. 86) for the complete details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.14.11. FAST Solution Files

This file type is valid only for triangular and tetrahedral meshes. Select **FAST Solution** from the **File Type** drop-down list and click the **Write...** button. A single file is written containing density, velocity, and total energy data.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See Exporting Data During a Transient Calculation (p. 86) for the complete details.

3.14.12. FieldView Unstructured Files

Select **Fieldview Unstructured** from the **File Type** drop-down list. Next, specify the cell zones and surfaces from which you want data exported by making selections in the **Cell Zones** or **Surfaces** lists (you must select at least one zone or surface); note that by default, all of the cell zones will be exported. Then select the scalars you want to write in the **Quantities** list. Finally, click the **Write...** button to open the **Select File** dialog box, which you can use to save a file that contains the specified function(s) for the specified cell zones and associated boundary zones or surfaces.

The following files are written:

- A binary file (for example, `file.fvuns`) containing coordinate and connectivity information and specified scalar function data. In parallel, each partition writes to this file with a grid for each cell zone or surface that is exported. This allows parallel postprocessing in FieldView.
- A regions file (for example, `file.fvuns.fvreg`) containing information about the cell zones or surfaces and the frame of reference. It also includes information about which grids make up the various cell zones or surfaces.

The cell zone and surface information includes the names of the cell zones/surfaces along with the mesh numbers. For the moving frame of reference, the regions file contains information about the origin, the axis of rotation and the rotation speed. Volume data is written using the absolute frame of reference.

If you are running multiple steady-state solutions on the same mesh, you can export only the data files and avoid the repeated writing of the mesh file by using the following TUI command for surfaces and cell zones:

`file/export/fieldview-unstruct-data` *file_name* `(` *list_of_surfaces* `) (` *list_of_zones* `)` *list_of_scalars* `q`

where

- *file_name* specifies the name (without the extension) of the file that you want to write.
- *list_of_surfaces* specifies the list of surfaces (separated by spaces, that is, no commas) from which you want data exported. If you want to specify all of the surfaces, enter `*` within the parentheses.
- *list_of_zones* specifies the list of cell zones (separated by spaces, that is, no commas) from which you want data exported. If you want to specify all of the cell zones, enter `*` within the parentheses.
- *list_of_scalars* specifies the list of cell functions (separated by spaces, that is, no commas) that you want to write to the exported file. The `q` input terminates the list. For example, the input `x-velocity cell-zone q` will select *x* velocity and the cell volume.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To export only the data files for surfaces, use the following TUI command:

`file/export/fieldview-unstruct-surfaces 2` *file_name* `(` *list_of_surfaces* `)` *list_of_scalars* `q`

where

- 2 specifies that you want to write a Fieldview unstructured results only file.
- *file_name* specifies the name (without the extension) of the file that you want to write.
- *list_of_surfaces* specifies the list of surfaces (separated by spaces, that is, no commas) from which you want data exported. If you want to specify all of the surfaces, enter `*` within the parentheses.
- *list_of_scalars* specifies the list of cell functions (separated by spaces, that is, no commas) that you want to write to the exported file. The `q` input terminates the list. For example, the input `x-velocity q` will select only the *x* velocity.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See Exporting Data During a Transient Calculation (p. 86) for the complete details.

3.14.13. I-deas Universal Files

Important

If you intend to export data to I-deas, ensure that the mesh does not contain pyramidal elements, as these are currently not supported by I-deas.

Select **I-deas Universal** from the **File Type** drop-down list. Select the surface(s) for which you want to write data in the **Surfaces** list. If no surfaces are selected, the entire domain is exported. You can specify which scalars you want in the **Quantities** list.

You have the option of selecting loads to be included in the exported file. When the **Energy Equation** is enabled in the Energy Dialog Box (p. 2175), you can choose the loads based on the kind of finite element analysis you intend to undertake. By selecting **Structural** in the **Analysis** list, you can select the following **Structural Loads**: **Force**, **Pressure**, and **Temperature**. By selecting **Thermal** in the **Analysis** list, you can select the following **Thermal Loads**: **Temperature**, **Heat Flux**, and **Heat Trans Coeff**.

These loads have the following limitations:

- When the **Energy Equation** is disabled, only the **Structural Loads** options of **Force** and **Pressure** are available.
- Loads are written only on boundary walls when the entire domain is exported (that is, if no **Surfaces** are selected).

Important

For non-stationary reference zones, all the velocities are exported to I-deas Universal as velocities relative to the selected reference zone. See the informational note in Data Explorer Files (p. 76) for further details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Click the **Write...** button to save the file, using the **Select File** dialog box. A single file is written containing coordinates, connectivity, optional loads, zone groups, velocity, and selected scalars.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See Exporting Data During a Transient Calculation (p. 86) for the complete details.

3.14.14. NASTRAN Files

Select **NASTRAN** from the **File Type** drop-down list. Select the surface(s) containing the data you want to export in the **Surfaces** list. If you do not select any surfaces, the entire domain is exported. You can specify which scalars you want in the **Quantities** list.

Note

If you are exporting data on a polyhedral mesh, only zone surfaces will be available for selection (no volumetric data or user-defined surfaces).

You have the option of selecting loads to be included in the exported file. When the **Energy Equation** is enabled in the Energy Dialog Box (p. 2175), you can choose the loads based on the kind of finite element analysis you intend to undertake. By selecting **Structural** in the **Analysis** list, you can select the following **Structural Loads**: **Force**, **Pressure**, and **Temperature**. By selecting **Thermal** in the **Analysis** list, you can select the following **Thermal Loads**: **Temperature**, **Heat Flux**, and **Heat Trans Coeff**.

These loads have the following limitations :

- When the **Energy Equation** is disabled, only the **Structural Loads** options of **Force** and **Pressure** are available.
- Loads are written only on boundary walls when the entire domain is exported (that is, if no **Surfaces** are selected).

Click the **Write...** button to save the file, using the **Select File** dialog box. A single file (for example, `file.bdf`) is written containing coordinates, connectivity, optional loads, zone groups, and velocity. Pressure is written as PLOAD4, and heat flux is written as QHBDYE data. If you select wall zones in the **Surfaces** list, nodal forces are written for the walls. When data is written for the heat transfer coefficient, it is based on the wall faces rather than the nodes.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. For details, see Exporting Data During a Transient Calculation (p. 86).

3.14.15. PATRAN Files

Select **PATRAN** from the **File Type** drop-down list. Select the surface(s) for which you want to write data in the **Surfaces** list. If you do not select any surfaces, the entire domain is exported. You can specify which scalars you want in the **Quantities** list.

You have the option of selecting loads to be included in the exported file. When the **Energy Equation** is enabled in the Energy Dialog Box (p. 2175), you can choose the loads based on the kind of finite element analysis you intend to undertake. By selecting **Structural** in the **Analysis** list, you can select the following **Structural Loads**: **Force**, **Pressure**, and **Temperature**. By selecting **Thermal** in the **Analysis** list, you can select the following **Thermal Loads**: **Temperature**, **Heat Flux**, and **Heat Trans Coeff**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These loads have the following limitations :

- When the **Energy Equation** is disabled, only the **Structural Loads** options of **Force** and **Pressure** are available.
- Loads are written only on boundary walls when the entire domain is exported (that is, if no **Surfaces** are selected).

Click the **Write...** button to save the file, using the **Select File** dialog box. A neutral file (for example, `file.out`) is written containing coordinates, connectivity, optional loads, zone groups, velocity, and selected scalars. Pressure is written as a distributed load. If wall zones are selected in the **Surfaces** list, nodal forces are written for the walls. The PATRAN result template file (for example, `file.res_tmpl`) is written, which lists the scalars present in the nodal result file (for example, `file.rst`).

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. For details, see Exporting Data During a Transient Calculation (p. 86).

3.14.16. RadTherm Files

The option to export a RadTherm file type is available only when the **Energy Equation** is enabled in the Energy Dialog Box (p. 2175). Select **RadTherm** from the **File Type** drop-down list and select the surface(s) for which you want to write data in the **Surfaces** list. If no surfaces are selected, the entire domain is exported. If the mesh contains polyhedral cells, ANSYS Fluent will export only boundary data. No volume data will be exported.

Select the method of writing the heat transfer coefficient (**Heat Transfer Coefficient**), which can be **Flux Based** or, if a turbulence model is enabled, **Wall Function** based. The flux-based heat transfer coefficient is calculated using Equation 6.118 (p. 382). If no turbulence model is enabled, the wall-function based heat transfer coefficient is calculated as k/h, where k is the thermal conductivity, and h is the cell height of the wall-adjacent cell.

Click the **Write...** button to save the file, using the **Select File** dialog box. A PATRAN neutral file (for example, `file.neu`) is written containing element velocity components (that is, the element that is just touching the wall), heat transfer coefficients, and temperatures of the wall for any selected wall surface. If the wall is one-sided, the data is written for one side of the wall. If the wall is two-sided (a wall-wall shadow pair), the values are written only for the original wall face, not for the shadow face (which is a duplicate).

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See Exporting Data During a Transient Calculation (p. 86) for the complete details.

Note

If no turbulence model is enabled, the **Wall Function** based Heat Transfer Coefficient is equal to k/h, where k is the thermal conductivity, and h is the cell height of the wall-adjacent cell.

3.14.17. Tecplot Files

Select **Tecplot** from the **File Type** drop-down list and choose the surface(s) for which you want to write data in the **Surfaces** list. If no surfaces are selected, the data is written for the entire domain. Select the variable(s) for which data is to be saved in the **Quantities** list.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Click the **Write...** button to save the file, using the **Select File** dialog box. A single file is written containing the coordinates and scalar functions in the appropriate tabular format.

Important

- The **utility fe2ram** can import Tecplot files only in **FEPOINT** format.
- If you intend to postprocess ANSYS Fluent data with Tecplot, you can either export data from ANSYS Fluent and import it into Tecplot, or use the Tecplot ANSYS Fluent Data Loader included with your Tecplot distribution. The data loader reads native ANSYS Fluent case and data files directly. If you are interested in this option, contact Tecplot, Inc. for assistance or visit www.tecplot.com.

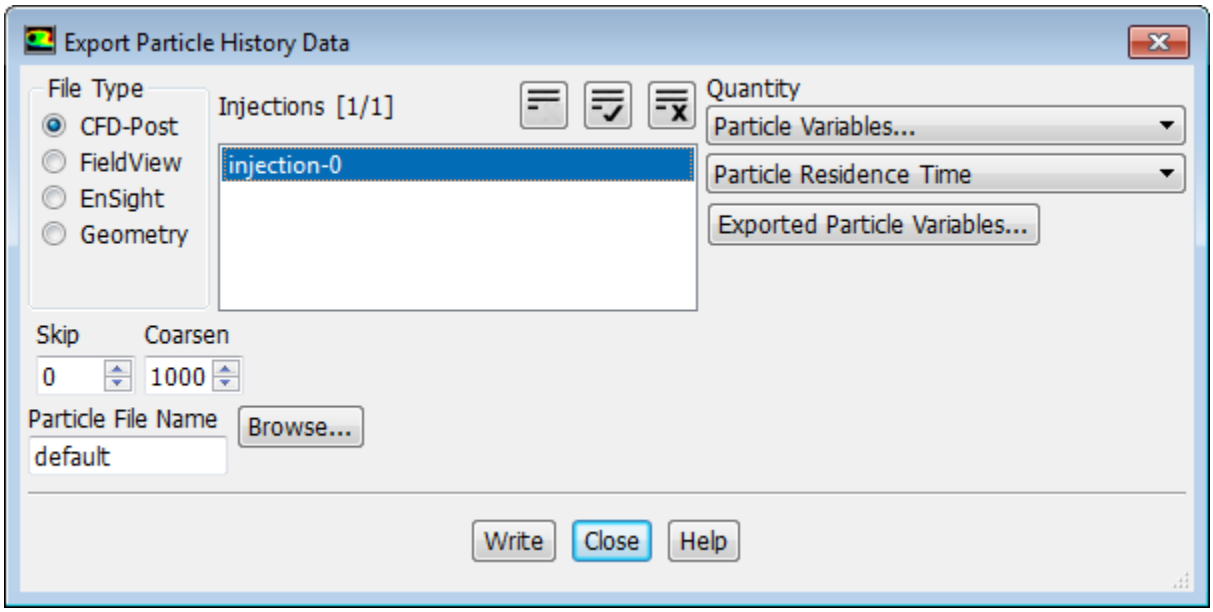
You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See Exporting Data During a Transient Calculation (p. 86) for the complete details.

3.15. Exporting Steady-State Particle History Data

Particle history data can be exported for steady-state solutions or for single transient particle steps by selecting the **Particle History Data...** option under the **File/Export** ribbon tab and performing the steps described in this section. For details about exporting particle history data automatically during transient simulations, see Creating Automatic Export Definitions for Transient Particle History Data (p. 89).

File → Export → Particle History Data...

Figure 3.6: The Export Particle History Data Dialog Box



1. Specify the **File Type** you want to export by selecting one of the following:
   - **CFD-Post** for the CFD-Post compatible format
   - **FieldView** for the FIELDVIEW format

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **EnSight** for the EnSight format
- **Geometry** for the `.ibl` format (not available when **Unsteady Particle Tracking** is enabled in the **Discrete Phase Model** dialog box)

Important

If you plan to export particle data to EnSight, you should first verify that you have already written the files associated with the EnSight Case Gold file type by using the **File/Export/Solution Data...** ribbon tab option (see EnSight Case Gold Files (p. 76)).

Select the predefined injections that are the source of the particles from the **Injections** selection list. See Creating and Modifying Injections (p. 1284) for details about creating injections.

2. Select the particle variables contained in the export file by clicking the **Exported Particle Variables...** button and selecting the variables appearing in the **Reporting Variables** dialog box (Figure 23.46: The Reporting Variables Dialog Box (p. 1348)), as described in Reporting of Current Positions for Unsteady Tracking (p. 1349).
3. If you have added the **Color by** variable in the **Reporting Variables** dialog box, select an appropriate category and variable under **Quantity** for the particle data to be exported.
4. If your exported particle history file is too large to postprocess because there are too many tracks or particles written to the file, you can reduce the number of particle tracks by increasing the **Skip** value.
5. To control the exported file size, the number of points of the particle trajectories can be reduced using the **Coarsen** value. This is only valid for steady-state particle trajectories.
6. Enter the name (and folder path, if you do not want it to be written in the current folder) for the exported particle data file in the **Particle File Name** text box. Alternatively, you can specify it through the **Select File** dialog box, which is opened by clicking the **Browse...** button.
7. If you selected **EnSight** under **File Type**, you should specify the **EnSight Encas File Name**. Use the **Browse...** button to select the `.encas` file that was created when you exported the file with the **File/Export/Solution Data...** ribbon tab option. The selected file will be modified and renamed as a new file that contains information about all of the related particle files that are generated during the export process (including geometry, velocity, scalars, particle and particle scalar files).

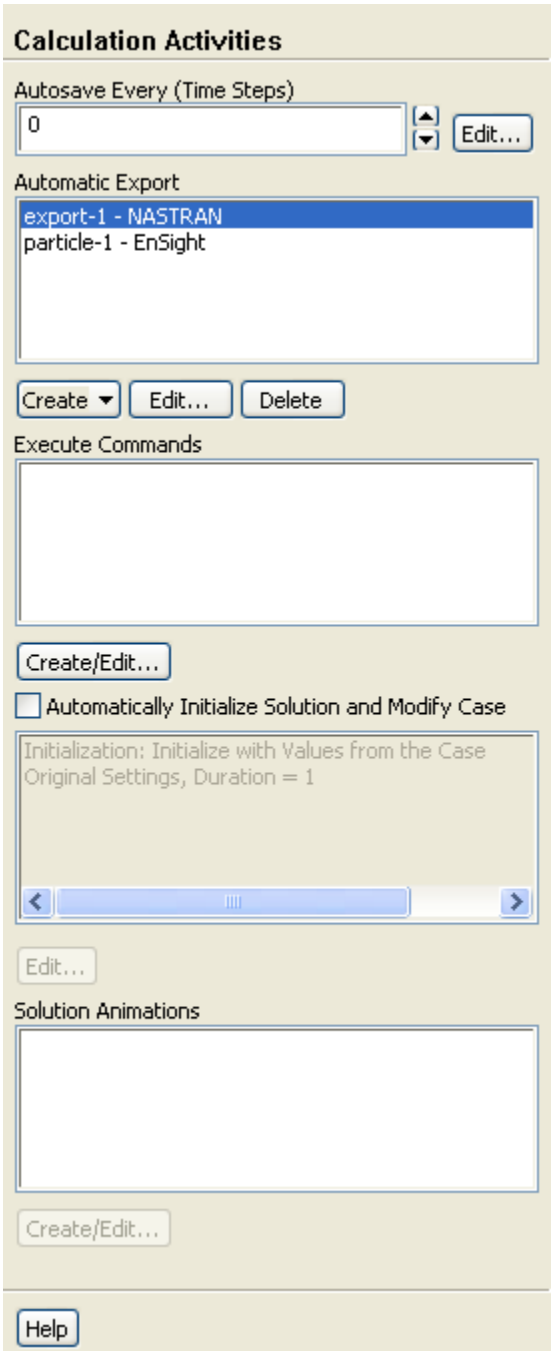
   The name of the new file will be the root of the original file with `.new` appended to it (for example, if `test.encas` is selected, a file named `test.new.encas` will be written). It is this new file that should be read into EnSight. If you do not specify a **EnSight Encas File Name**, then you must create an appropriate `.encas` file manually.
8. If you selected **EnSight** under **File Type**, and you are exporting steady-state particle tracks, enter the **Number of Particle Time Steps**.
9. Click **Write** to export the particle history data. If you selected **EnSight** under **File Type**, data files will be written in both `.mpg` and `.mscl` formats.
10. Click **Close** to close the dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.16. Exporting Data During a Transient Calculation

Before you run a transient flow solution, you can set up the case file so that solution data and particle history data is exported as the calculation progresses. This is accomplished by creating automatic export definitions using the Calculation Activities Task Page (p. 2525) (Figure 3.7: The Calculation Activities Task Page (p. 86)), as described in the following sections.

Figure 3.7: The Calculation Activities Task Page



The names of the automatic export definitions you create are displayed in the **Automatic Export** selection list, along with the format in which it will be exported. You can edit or delete the definition by selecting a definition in the list and clicking the **Edit...** or **Delete** button,.

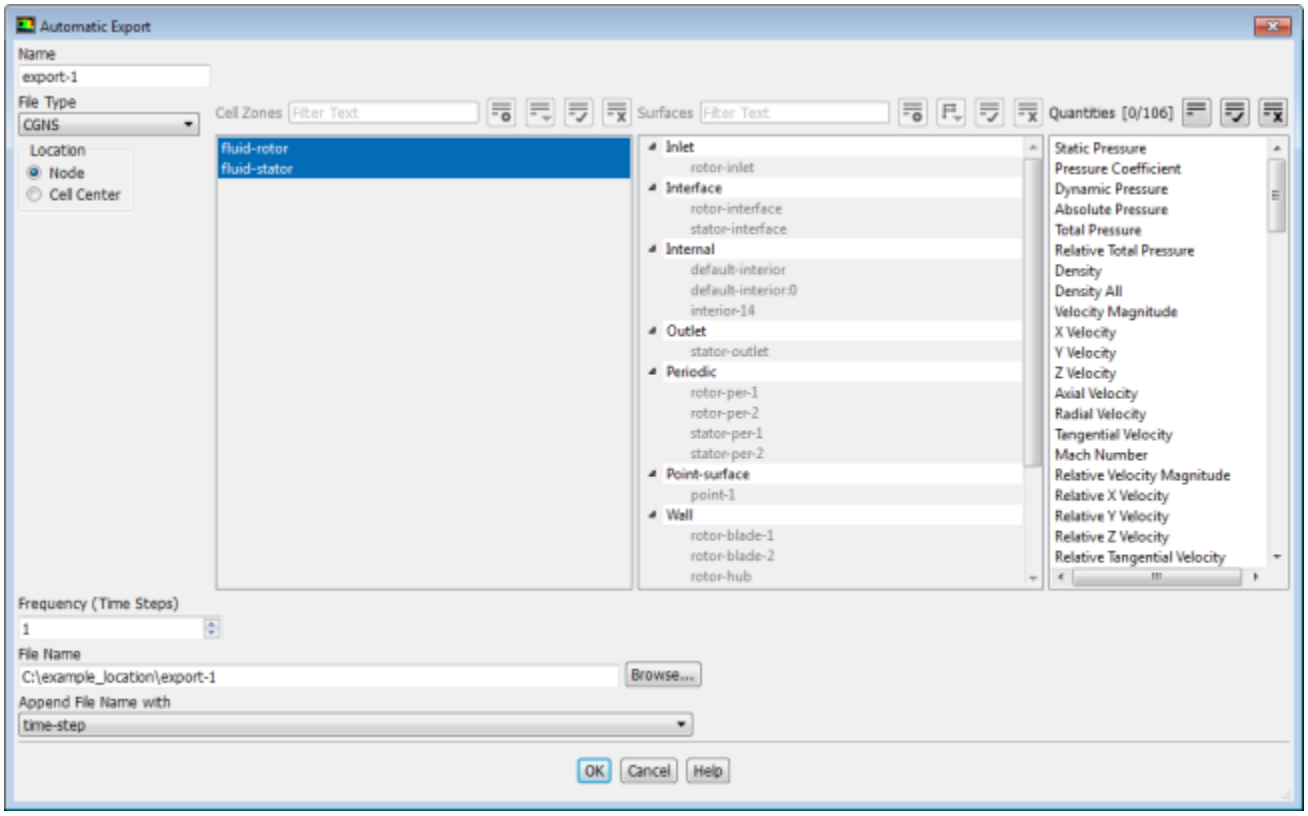
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For additional information, see the following sections:
3.16.1. Creating Automatic Export Definitions for Solution Data
3.16.2. Creating Automatic Export Definitions for Transient Particle History Data

3.16.1. Creating Automatic Export Definitions for Solution Data

To create an automatic export definition for solution data, begin by making sure that **Transient** is selected for **Time** in the General Task Page (p. 2160). Next, click the **Create** button under the **Automatic Export** selection list in the **Calculation Activities** task page (a drop-down list will appear). Select **Solution Data Export...** from the drop-down list to open the **Automatic Export** dialog box (Figure 3.8: The Automatic Export Dialog Box (p. 87)).

Figure 3.8: The Automatic Export Dialog Box



Then perform the following steps:

1. Enter a name for the automatic export definition in the **Name** text box. This is the name that will be displayed in the **Automatic Export** selection list in the **Calculation Activities** task page.

2. Define the data to be exported by making selections in the relevant group boxes and selection lists: **File Type**, **Cell Zones**, **Surfaces**, **Interior Zone Surfaces**, **Quantities**, **Analysis**, **Structural Loads**, **Thermal Loads**, **Location**, **Delimiter**, **Format**, and **Heat Transfer Coefficient**. See ABAQUS Files (p. 71) – Tecplot Files (p. 83) for details about the specific options available for the various file types.

3. Set the **Frequency** at which the solution data will be exported during the calculation. If you enter `10` in the **Frequency** text box, for example, a file will be written after every 10 time steps.

4. If you selected **EnSight Case Gold** from the **File Type** drop-down list, the **Separate Files for Each Time Step** option allows you to specify that separate files are written at the prescribed time steps. This option is enabled by default and is the recommended practice, as it ensures that all of the data is not lost if there

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

is a disruption to the calculation (for example, from a network failure) before it is complete. If you choose to disable this option, all of the data for the `.scl1` and `.vel` files will be combined into a single file for each.

5. If you selected **CFD-Post Compatible** from the **File Type** drop-down list, by default ANSYS Fluent will save a case (`.cas`) file with every `.cdat` file (that is, at the specified **Frequency**). You can change the criteria for when case files are saved by disabling the **Write Case File Every Time** option; then, ANSYS Fluent will save case files according to the settings specified by the following text command:

   `file` → `transient-export` → `settings` → `cfd-post-compatible`

   By default, the `cfd-post-compatible` text command is set to save a case file only if ANSYS Fluent detects that the mesh or case file has been modified.

   Note that regardless of the settings, only a single `.cst` file will be written when exporting during a transient calculation.

   For more information about the **CFD-Post Compatible** file type, go to ANSYS CFD-Post-Compatible Files (p. 74).

6. Specify how the exported files will be named. Every file saved will begin with the characters entered in the **File Name** text box (note that a file extension is not necessary). You can specify a folder path if you do not want it written in the current folder. The **File Name** can also be specified through the **Select File** dialog box, which is opened by clicking the **Browse...** button.

   Next, make a selection in the **Append File Name with** drop-down list, to specify that the **File Name** be followed by either the time step or flow time at which it was saved. Note that this selection is not available when exporting to EnSight. When **EnSight Case Gold** is selected from the **File Type** drop-down list, the time step is always appended if the **Separate Files for Each Time Step** option is enabled; otherwise, no digits are appended.

   When appending the file name with the flow time, you can specify the number of decimal places that will be used by making an entry in the **Decimal Places in File Name** text box. By default, six decimal places will be used.

7. Click **OK** to save the settings for the automatic export definition.

For details about general limitations for exporting solution data and the manner in which it is exported, see Exporting Solution Data (p. 69).

Important

- If the files that are exported during multiple transient simulations are to be used as a set, you should run all of the simulations on the same platform, using the same number of processors. This ensures that all of the files are compatible with each other.

  Note

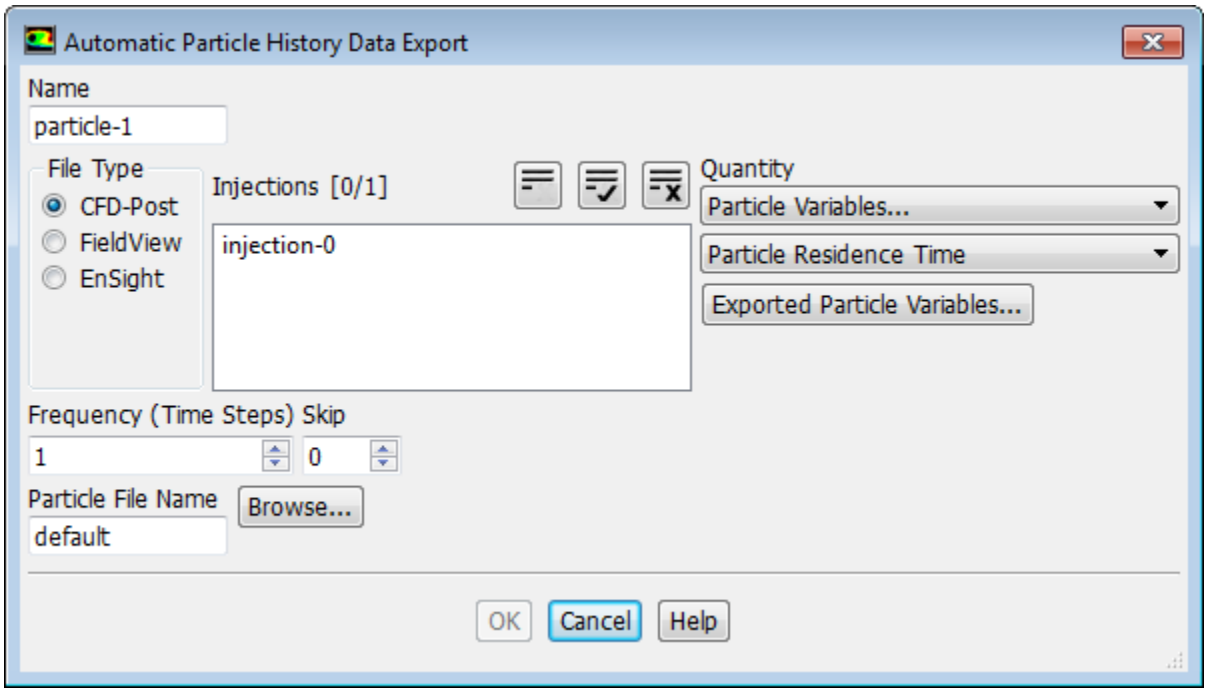
  If you selected **EnSight Case Gold** from the **File Type** drop-down list, ANSYS Fluent does not support exporting data files to EnSight during a transient calculation in which a new cell zone or surface is created after the calculation has begun (as can be the case for an in-cylinder simulation, for example).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.16.2. Creating Automatic Export Definitions for Transient Particle History Data

To create an automatic export definition for particle history data, begin by making sure that **Unsteady Particle Tracking** is selected in the **Discrete Phase Model** dialog box. Next, click the **Create** button under the **Automatic Export** selection list in the **Calculation Activities** task page (a drop-down list will appear). Select **Particle History Data Export...** from the drop-down list to open the **Automatic Particle History Data Export** dialog box (Figure 3.9: The Automatic Particle History Data Export Dialog Box (p. 89)).

Figure 3.9: The Automatic Particle History Data Export Dialog Box



Then perform the following steps:

1. Enter a name for the automatic export definition in the **Name** text box. This is the name that will be displayed in the **Automatic Export** selection list in the **Calculation Activities** task page. Make sure that the chosen name is not already used for another **Automatic Export**.

2. Choose the **File Type** you want to export by selecting one of the following:

   - **CFD-Post** for the CFD-Post compatible format
   - **FieldView** for the FIELDVIEW format
   - **EnSight** for the EnSight format

   Important

   If you plan to export particle data to EnSight, you should first set up an automatic export definition so that solution data is also exported to EnSight during this calculation (see Creating Automatic Export Definitions for Solution Data (p. 87)). As described in the steps

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

that follow, some of the settings must correspond between the two automatic export definitions.

3. Select the predefined injections that are the source of the particles from the **Injections** selection list. See Creating and Modifying Injections (p. 1284) for details about creating injections.

4. Select the particle variables contained in the export file by clicking the **Exported Particle Variables...** button and selecting the variables appearing in the **Reporting Variables** dialog box (Figure 23.46: The Reporting Variables Dialog Box (p. 1348)), as described in Reporting of Current Positions for Unsteady Tracking (p. 1349).

5. If you have added the **Color by** variable in the **Reporting Variables** dialog box, select an appropriate category and variable under **Quantity** for the particle data to be exported.

6. Set the **Frequency** at which the particle history data will be exported during the calculation. If you enter `10` in the **Frequency** text box, for example, a file will be written after every 10 time steps for transient flow cases or 10 DPM Iterations for steady flow cases.

7. If your exported particle history file is too large to postprocess because there are too many tracks or particles written to the file, you can reduce the number of particle tracks by increasing the **Skip** value.

8. If you selected **EnSight Case Gold** for the **File Type**, the **Separate Files for Each Time Step** option allows you to specify that separate files are written at the prescribed time steps. This option is enabled by default and is the recommended practice, as it ensures that all of the data is not lost if there is a disruption to the calculation (for example, from a network failure) before it is complete. If you choose to disable this option, all of the data for the `.mscl` and `.mpg` files will be combined into a single file for each.

   Important

   The setting for the **Separate Files for Each Time Step** option should be the same (that is, enabled or disabled) as that of the automatic export definition you set up to export solution data to EnSight during this calculation. For details, see Creating Automatic Export Definitions for Solution Data (p. 87).

9. Enter the name (and folder path, if you do not want it to be written in the current folder) for the exported particle data file in the **Particle File Name** text box. Alternatively, you can specify it through the **Select File** dialog box, which is opened by clicking the **Browse...** button. Make sure the file name is different from other existing **Automatic Export** definitions to avoid overwriting data.

10. If you selected **EnSight** under **File Type**, you should specify the **EnSight Encas File Name**. Enter the same name (and folder path, if necessary) that you entered in the **File Name** text box when you set up the automatic export definition for exporting solution data to EnSight during this calculation. (For details, see Creating Automatic Export Definitions for Solution Data (p. 87)). The `.encas` file created during the solution data export will be modified and renamed as a new file that contains information about all of the related particle files that are generated after every time step during the export process (including geometry, velocity, scalars, particle and particle scalar files). The name of the new file will be the root of the original `.encas` file with `.new` appended to it (for example, if the solution data export creates `test.encas`, a

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

file named `test.new.encas` will be written for the particle data export). It is this new file that should be read into EnSight.

Important

If you do not specify an **EnSight Encas File Name**, you will have to manually create an appropriate `.encas` file.

11. Click **OK** to save the settings for the automatic export definition.

The particle data will be exported as it is generated during the transient calculation. If you selected **EnSight** under **File Type**, data files will be written in both `.mpg` and `.mscl` formats.

3.17. Exporting to ANSYS CFD-Post

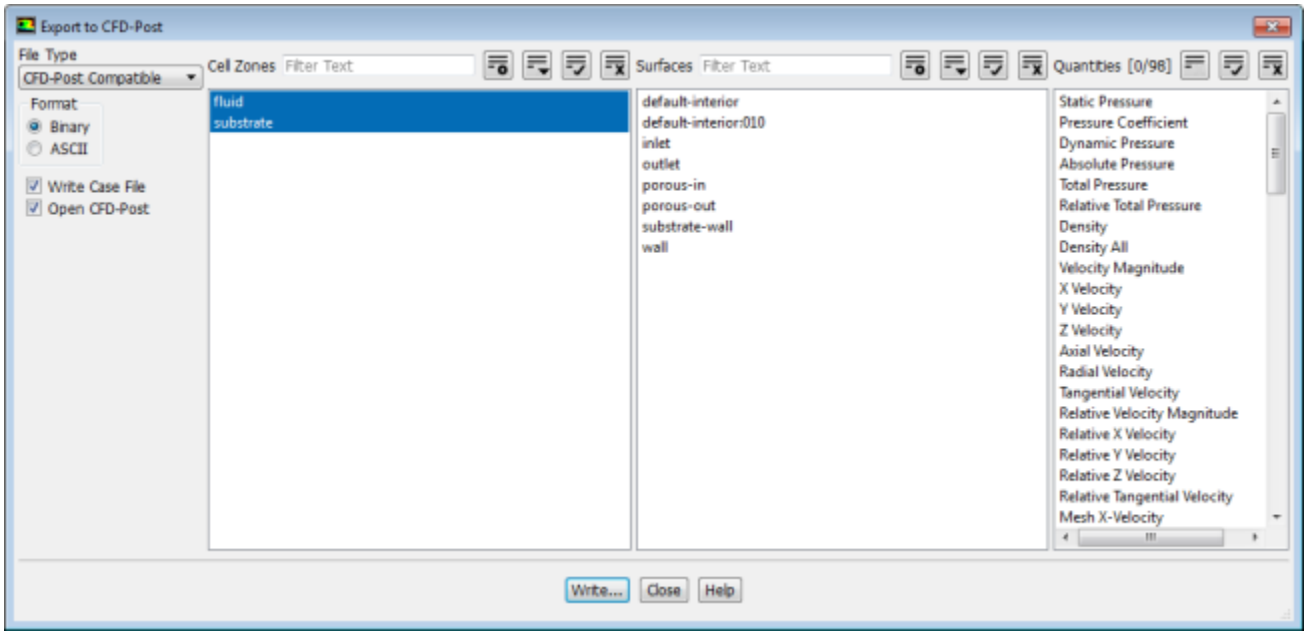
You can use the **Export to CFD-Post** dialog box (Figure 3.10: The Export to CFD-Post Dialog Box (p. 91)) to export the results from cell zones and/or surfaces to files that are compatible with ANSYS CFD-Post.

Important

When reading `.dat` files into ANSYS CFD-Post, some variables may show incorrect values in some conditions. You should use `.cdat` files or you may need to cross check these values between CFD-Post and Fluent.

File → Export to CFD-Post...

Figure 3.10: The Export to CFD-Post Dialog Box



The **Quantities** selection list contains the variables that you can choose to save for postprocessing. When selecting the variables, be sure to include any variable that was used to create an isosurface or clipped surface (as described in Isosurfaces (p. 1728) and Clipping Surfaces (p. 1730), respectively); otherwise, the surface will not be created properly if you try to read the exported state file in CFD-Post.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specify the format of the .cdat file by selecting either **Binary** or **ASCII** from the **Format** list. The advantage of the binary format is that it takes less time to load the exported data into ANSYS CFD-Post and requires less storage space. Note that the format for the .cst will always be ASCII.

By default, Fluent will write a case file (that is, .cas) along with the **CFD-Post Compatible** files. If you do not want such a case file for any reason (for example., to improve I/O performance or save disc space), disable the **Write Case File** option. Disabling this option is only recommended if you are performing multiple export operations for a case file that is not changing.

By default, the **Open CFD-Post** option is enabled so that the following actions will occur after the files are exported:

1. A CFD-Post session opens automatically.

2. The case and .cdat files are loaded in CFD-Post.

3. CFD-Post displays the results.

If you disable the **Open CFD-Post** option, you can open CFD-Post manually and use the **Load Results** item in the **File** drop-down menu to load the results files. For more information about this feature in CFD-Post, see the separate ANSYS CFD-Post manual.

Click the **Write...** button to save the files, using the **Select File** dialog box. A .cdat file is written, containing the specified variable data for the specified cell zones and all of the boundary zones. A state file (that is, .cst) is also written, which contains the following surfaces that you created in Fluent for postprocessing: point surfaces, line surfaces, plane surfaces, isosurfaces, and clipped surfaces (see Creating Surfaces and Cell Registers for Displaying and Reporting Data (p. 1709)).

Note

Imprinted surfaces defined in ANSYS Fluent are not available for postprocessing in ANSYS CFD-Post.

Important

When you read the .cdat file into ANSYS CFD-Post, the application will attempt to read in the case file that produced the data by looking in the folder for a .cas file with the same prefix. If the case file is not in that folder (e.g., if you disabled the **Write Case File** option when exporting), ANSYS CFD-Post will prompt you to specify the appropriate case file.

Important

Before loading the .cst file into ANSYS CFD-Post, make sure that the .cas and .cdat files are already loaded for the session.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See Exporting Data During a Transient Calculation (p. 86) for the complete details.

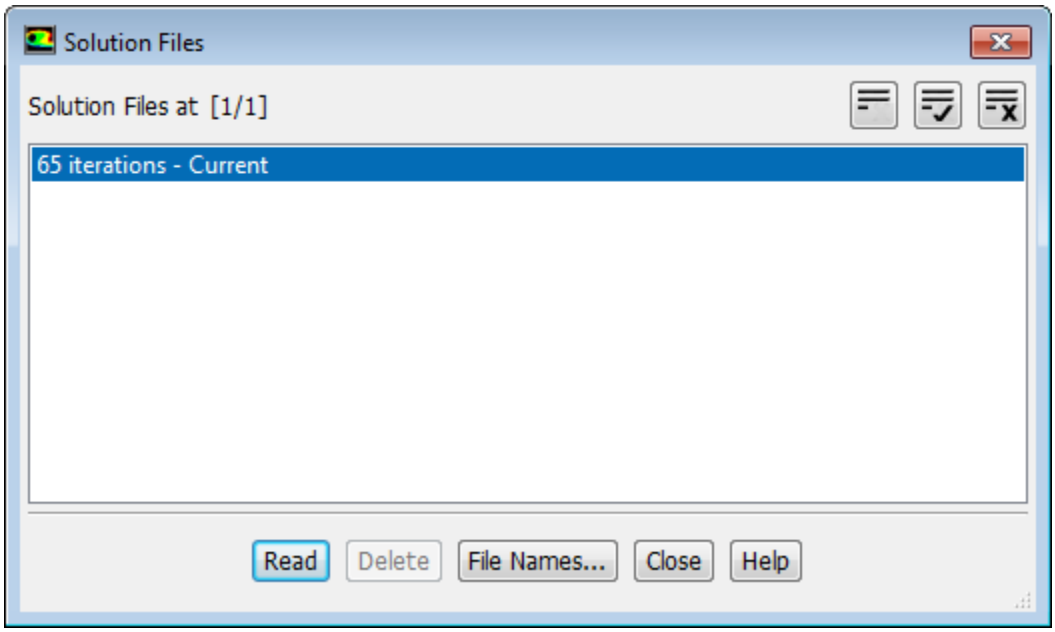
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.18. Managing Solution Files

You can manage your solution files effectively and efficiently using the **Solution Files** dialog box. Here, you can select previously saved files created using the **Autosave** dialog box and read or delete them.

File → Solution Files...

Figure 3.11: The Solution Files Dialog Box



The **Solution Files** dialog box (Figure 3.11: The Solution Files Dialog Box (p. 93)) lists all of the solution files that have been automatically saved. They are listed by iteration number or time step/flow time. For the file that is currently read in, the status of `current` will appear in the **Solution Files at** list. You can make any of the files in the list current by clicking the **Read** button. Note that if more than one file is selected, the **Read** button is disabled. When an earlier solution is made current, the solution files that were generated for a later iteration/time step will be removed from this list when the calculation continues.

You can delete solution files by selecting an entry in the list and clicking **Delete**. Note that a currently loaded solution file cannot be deleted, however multiple (non-current) files can be selected and deleted. If multiple files are selected and one of those files is a currently loaded solution file, clicking **Delete** will result in the current solution file being skipped.

You can click the **File Names...** button to obtain information about the solution files and the path of the associated files.

When running Fluent under Workbench, you can recover the latest case and data files from the system folder (`<project name>\dp0\FFF\Fluent\` for analysis system and `<project name>\dp0\FLU \Fluent\` for component system) by clicking **Recover Missing Solution...** The recovered files will appear in the **Solution Files at** list. You can use this option when the latest matching/compatible case and data files are missing from the **Solution Files at** list.

The **Solution Files** dialog box is particularly useful for reading in files that were saved during the autosave session, since case and data files may not necessarily have the same file name.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.19. Mesh-to-Mesh Solution Interpolation

ANSYS Fluent can interpolate solution data for a given geometry from one mesh to another, allowing you to compute a solution using one mesh (for example, hexahedral) and then change to another mesh (for example, hybrid) and continue the calculation using the first solution as a starting point.

Important

ANSYS Fluent does zeroth-order interpolation for interpolating the solution data from one mesh to another.

For additional information, see the following sections:

3.19.1. Performing Mesh-to-Mesh Solution Interpolation
3.19.2. Format of the Interpolation File

3.19.1. Performing Mesh-to-Mesh Solution Interpolation

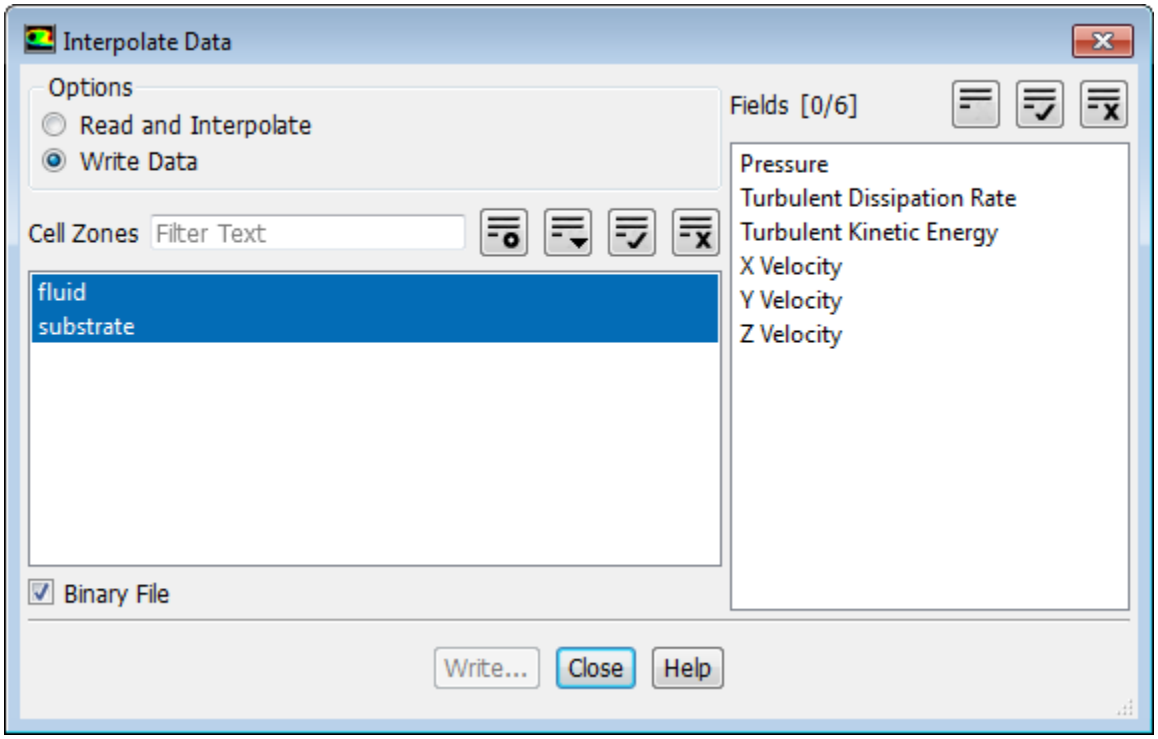
The procedure for mesh-to-mesh solution interpolation is as follows:

1. Set up the model and calculate a solution on the initial mesh.

2. Write an interpolation file for the solution data to be interpolated onto the new mesh, using the **Interpolate Data** dialog box (Figure 3.12: The Interpolate Data Dialog Box (p. 94)).

   File → Interpolate...

   Figure 3.12: The Interpolate Data Dialog Box



   a. Under **Options**, select **Write Data**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. In the **Cell Zones** selection list, select the cell zones for which you want to save data to be interpolated.

> **Note**
>
> If your case includes both fluid and solid zones, write the data for the fluid zones and the data for the solid zones to separate files.

c. Select the variable(s) for which you want to interpolate data in the **Fields** selection list. All ANSYS Fluent solution variables are available for interpolation.

d. Select the **Binary File** check box if you want a binary interpolation file to be generated.

> **Note**
>
> Writing a binary interpolation file is significantly faster and requires less memory than writing a text file.

e. Click **Write...** and specify the interpolation file name in the resulting **Select File** dialog box. The file format is described in Format of the Interpolation File (p. 96).

3. Set up a new case.

a. Read in the new mesh, using the appropriate ribbon tab item (**File/Read/** or **File/Import/**).

b. Define the appropriate models.

> **Important**
>
> Enable all of the models that were enabled in the original case. For example, if the energy equation was enabled in the original case and you forget to enable it in the new case, the temperature data in the interpolation file will not be interpolated.

c. Define the boundary conditions, material properties, and so on.

> **Important**
>
> An alternative way to set up the new case is to save the boundary conditions from the original model using the `write-settings` text command, and then read in those boundary conditions with the new mesh using the `read-settings` text command. See Reading and Writing Boundary Conditions (p. 56) for further details.

4. Read in the data to be interpolated.

File → Interpolate...

a. Under **Options**, select **Read and Interpolate**.

b. In the **Cell Zones** list, select the cell zones for which you want to read and interpolate data.

   If the solution has not been initialized, computed, or read, all zones in the **Cell Zones** list are selected by default, to ensure that no zone remains without data after the interpolation. If all zones already have data (from initialization or a previously computed or read solution), select a subset of the **Cell Zones** to read and interpolate data onto a specific zone (or zones).

c. Click the **Read...** button and specify the interpolation file name in the resulting **Select File** dialog box.

Important

If your case includes both fluid and solid zones, the two sets of data are saved to separate files. Hence perform these steps twice, once to interpolate the data for the fluid zones and once to interpolate the data for the solid zones.

---

5. Reduce the under-relaxation factors and calculate on the new mesh for a few iterations to avoid sudden changes due to any imbalance of fluxes after interpolation. Then increase the under-relaxation factors and compute a solution on the new mesh.

3.19.2. Format of the Interpolation File

An example of an interpolation file is shown below:

```
3
2
34800
3
x-velocity
pressure
y-velocity
(-0.068062
-0.0680413
...
```

The format of the interpolation file is as follows:

- The first line is the interpolation file version. It is 2 for files generated using ANSYS Fluent 12.0 through 14.0, 3 for text files generated using ANSYS Fluent 14.5, 4 for binary files generated using single precision ANSYS Fluent 14.5, and 5 for files generated using double precision ANSYS Fluent 14.5.
- The second line is the dimension (2 or 3).
- The third line is the total number of points.
- The fourth line is the total number of fields (temperature, pressure, and so on) included.
- Starting at the fifth line is a list of field names. To see a complete list of the field names used by ANSYS Fluent, enter the `display/contour` text command and view the available choices by pressing **Enter** at the `contours of>` prompt. The list depends on the models turned on.
- After the field names is a section for each list of x, y, and (in 3D) z coordinates for all the data points.
- At the end is a section for each list of the field values at all the points in the same order as their names. The number of coordinate and field points should match the number given in line 3.
- For version 3 interpolation files, the sections are bounded by "(" and ")".

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For version 4 and 5 interpolation files, the sections are bounded by "(" and "\nEnd of Binary Section 0)".
- The delimiters help skip a section if the associated model is not enabled. With version 2 interpolation files (where the delimiters do not exist), sections may not be skipped properly if the size of the field cannot be determined without enabling the associated model. This may result in incorrect interpolation of the subsequent field variables.

Important

An interpolation file written with ANSYS Fluent 14.5 and above is not readable in prior versions of ANSYS Fluent.

3.20. Mapping Data for Fluid-Structure Interaction (FSI) Applications

ANSYS Fluent allows you to map variables (for example, temperature, pressure) from the cell or face zones of an ANSYS Fluent simulation onto locations associated with a finite element analysis (FEA) mesh. The results are written to a file for inclusion into an FEA simulation. During this process, both the original and the new mesh can be viewed simultaneously. ANSYS Fluent maps the data using zeroth-order interpolation, and can write the output file in a variety of formats.

This capability is useful when solving fluid-structure interaction (FSI) problems, and allows you to perform further analysis on the solid portion of your model using FEA software. Mapping the data may be preferable to simply exporting the ANSYS Fluent data file (as described in Exporting Solution Data (p. 69)), since the meshes used in CFD analysis are typically finer than those used in finite element analysis.

For additional information, see the following sections:

3.20.1. FEA File Formats
3.20.2. Using the FSI Mapping Dialog Boxes

3.20.1. FEA File Formats

The FEA software types that are compatible with ANSYS Fluent's FSI mapping capability include ABAQUS, I-deas, ANSYS, NASTRAN, and PATRAN. For details about the kinds of files that can be read or written during this process, see Table 3.4: FEA File Extensions for FSI Mapping (p. 97).

Table 3.4: FEA File Extensions for FSI Mapping

Type	Input File	Output File
ABAQUS	`.inp`	`.inp`
I-deas	`.unv`	`.unv`
ANSYS	`.cdb`, `.neu`	`.cdb`
NASTRAN	`.bdf`	`.bdf`
PATRAN	`.neu`, `.out`, `.pat`	`.out`

3.20.2. Using the FSI Mapping Dialog Boxes

To begin the process of mapping ANSYS Fluent data, you must first create a mesh file that can be used as the input file in the steps that follow. The resolution of the mesh should be appropriate for your eventual finite element analysis. You are free to use the method and preprocessor of your choice in the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

creation of this file, but the end result must correspond to one of the entries in the Input File column of Table 3.4: FEA File Extensions for FSI Mapping (p. 97).

When creating the input file, note the following:

- While the input file may be scaled when it is read into ANSYS Fluent, the volumes or surfaces on which the data is to be mapped must otherwise be spatially coincident with their counterparts in the ANSYS Fluent simulation.
- ANSYS Fluent can map volume and surface data only for 3D cases; data mapping is not supported for 2D cases since data mapping for edges is not supported.
- The input file can be only a portion of the overall FEA model (that is, you can exclude the parts of the model on which you are not mapping ANSYS Fluent data). When this is the case, note that the numbering of the nodes and elements in the input file must match the numbering of the nodes and elements in the complete file you will use for your finite element analysis.

Next, read a case file in ANSYS Fluent and make sure data is available for mapping, either by running the calculation or by reading a data file.

Finally, perform the following steps to generate an output file in which the ANSYS Fluent data has been mapped to the mesh of the input file:

1. Open the ANSYS Fluent dialog box that is appropriate for the zones from which the data is to be taken. If the data you are mapping is from a volume (for example, the cell zone of a solid region), open the **Volume FSI Mapping** dialog box using the **File/FSI Mapping/Volume...** ribbon tab item (Figure 3.13: The Volume FSI Mapping Dialog Box for Cell Zone Data (p. 99)). If instead the data is from a surface (for example, a face boundary zone), open the **Surface FSI Mapping** dialog box using the **File/FSI Mapping/Surface...** ribbon tab item (Figure 3.14: The Surface FSI Mapping Dialog Box for Face Zone Data (p. 99)).

   File → FSI Mapping → Volume...

   or

   File → FSI Mapping → Surface...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.13: The Volume FSI Mapping Dialog Box for Cell Zone Data

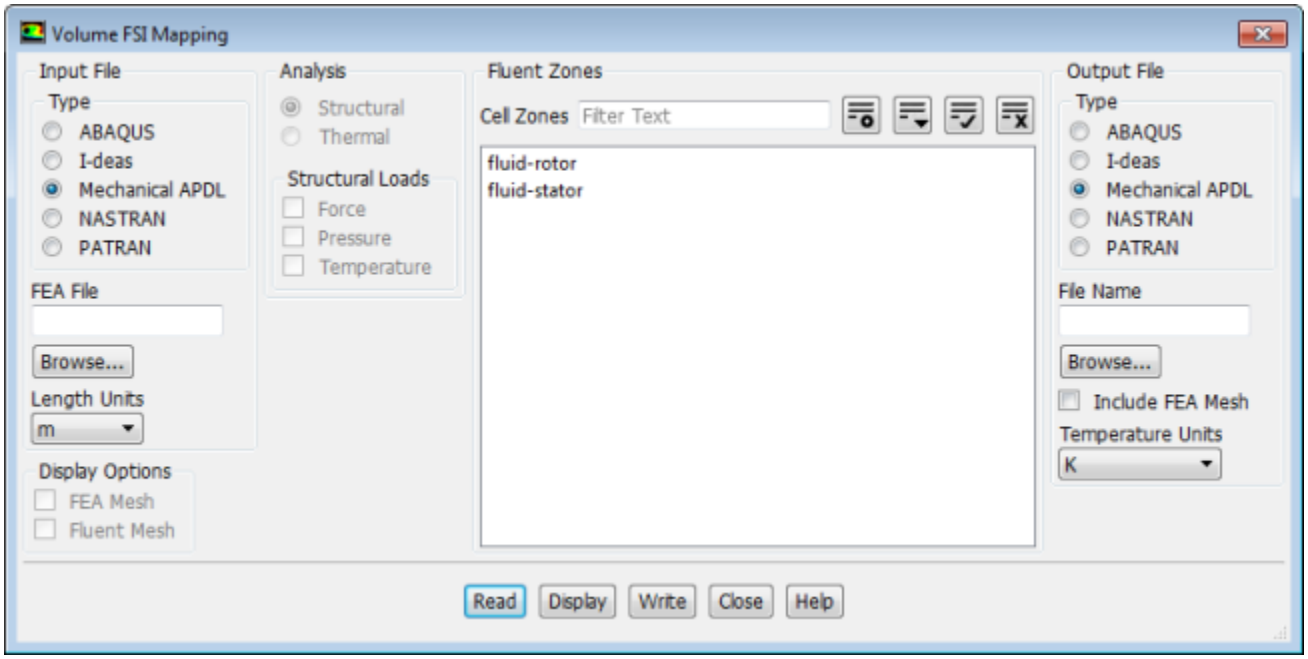
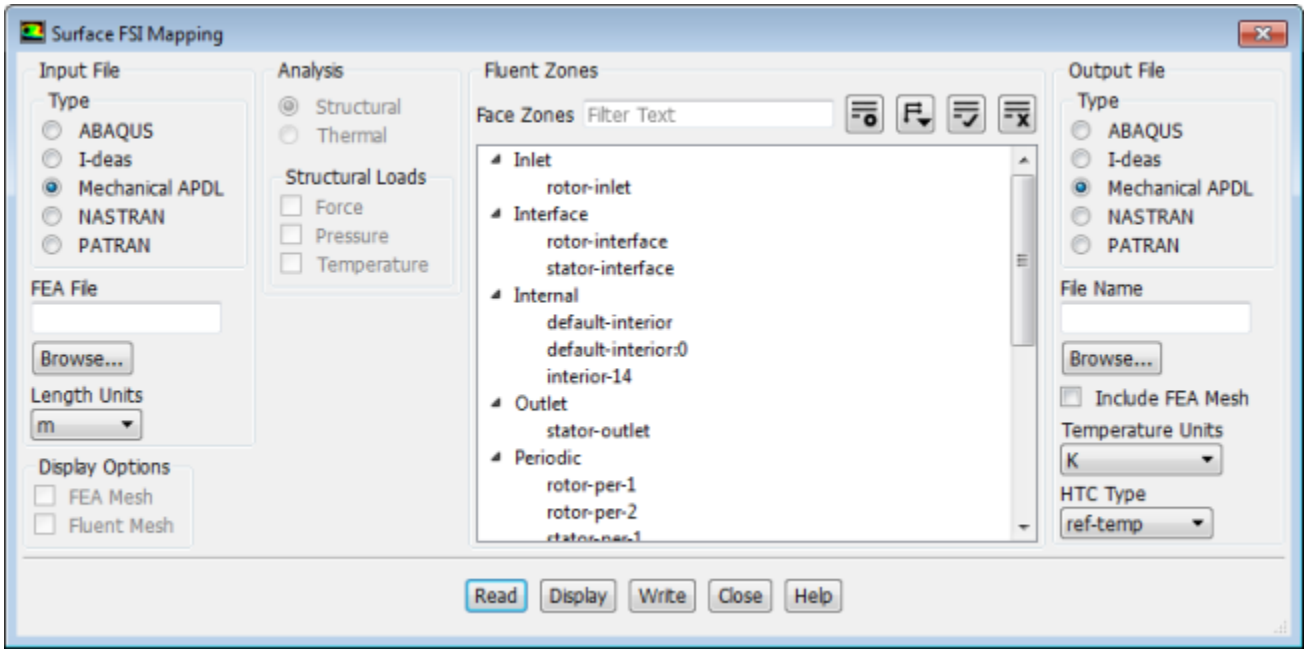


Figure 3.14: The Surface FSI Mapping Dialog Box for Face Zone Data



2. Specify the parameters of the input file and read it into ANSYS Fluent.

   a. Select the format of the input file from the **Type** list in the **Input File** group box, based on the FEA software with which it is associated. The choices include:

      - **ABAQUS**
      - **I-deas**
      - **Mechanical APDL**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **NASTRAN**
- **PATRAN**

For a list of the file extensions associated with these types, see the Input File column of Table 3.4: FEA File Extensions for FSI Mapping (p. 97).

b. Enter the name and extension (along with the folder path, if it is not in the current folder) of the input file in the **FEA File** text-entry box. Alternatively, you can specify it through the **Select File** dialog box, which is opened by clicking the **Browse...** button.

c. Specify the length units that were used in the creation of the input file by making a selection from the **Length Units** drop-down menu. This ensures that the input file is scaled appropriately relative to the ANSYS Fluent file.

d. Click the **Read** button to read the input file into memory.

Important

Note that the input file will only be held in memory until the output file is written, or until the FSI mapping dialog box is closed.

---

3. Display the meshes so that you can visually verify that the input file is properly scaled and aligned with the ANSYS Fluent mesh file.

a. Make sure that the **FEA Mesh** and **Fluent - Mesh** options are selected in the **Display Options** group box. Note that you can disable either of these options if you want to examine one of the meshes independently.

b. Click the **Display** button to display the meshes in the graphics window.

Important

For the ANSYS Fluent mesh, only the zones selected in the **Fluent Zones** group box will be displayed—in this case, the default selections. If the default zones are not appropriate, you should redisplay the meshes after you make your zone selections in a later step.

---

4. Specify the type of data variables to be mapped.

a. Select either **Structural** or **Thermal** in the **Analysis** group box. Your selection should reflect the kind of further analysis you intend to pursue, and will determine what variables are available for mapping.

b. Enable the variables you want to map in the **Structural Loads** or **Thermal Loads** group box. When mapping volume data, you can enable only **Temperature**. When mapping surface data, you can enable

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Force, **Pressure**, and **Temperature** for structural analysis, or **Temperature**, **Heat Flux**, and **Heat Trans Coeff** for thermal analysis.

Important

Note that the **Energy Equation** must be enabled in the **Energy** dialog box if you want to map temperature for a structural analysis or any variable for a thermal analysis.

5. Select the zones that contain the data to be mapped in the **FIluent Zones** group box. You can select individual zones in the **Cell Zones** or **Face Zones** selection lists, or select all zones of a particular type in the **Zone Type** selection list. If you modify the default selections, you should display the meshes again, as described previously.

   Important

   - Note that all wall zones in the **Face Zones** selection list are selected by default in the **Surface FSI Mapping** dialog box, and this includes the shadow walls created for two-sided walls. If your ANSYS Fluent file contains a wall/shadow pair (for example, separating a solid zone from a fluid zone), you should make sure that only the correct wall or shadow of the pair is selected.
   - Inlet zones do not have heat transfer coefficient data, and so any attempts to map this combination will be ignored.

6. Specify the parameters of the output file and write it.

   a. Select the format of the output file from the **Type** list in the **Output File** group box, based on the software with which you plan to perform your finite element analysis. The choices in this list are the same as those for the input file type. Note that you can select an output file type that is different from the input file type.

      For details about the file extensions associated with the various types of output files, see the Output File column of Table 3.4: FEA File Extensions for FSI Mapping (p. 97).

   b. Enter the name (with the folder path, if appropriate) of the output file in the **File Name** text-entry box. Alternatively, you can specify it through the **Select File** dialog box, which is opened by clicking the **Browse...** button.

   c. To include additional FEA information like node/element information in the exported output file, enable **Include FEA Mesh**. By default, this option is disabled and therefore, only the selected boundary condition values are exported.

   d. When mapping temperature for a structural analysis or any variable for a thermal analysis, make a selection in the **Temperature Units** drop-down menu. Table 3.5: Units Associated with the Temperature Units Drop-Down List Selections (p. 101) shows the units for the mapped variables, depending on the **Temperature Units** selection.

Table 3.5: Units Associated with the Temperature Units Drop-Down List Selections

Temperature Units Selection	Temperature	Heat Flux	Heat Transfer Coefficient
K	K	W/m^2	W/m^2-K

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Temperature Units Selection	Temperature	Heat Flux	Heat Transfer Coefficient
C	$^{\circ}$C	W/m^2	W/m^2- $^{\circ}$C
F	$^{\circ}$F	BTU/ft^2-hr	BTU/ft^2-hr- $^{\circ}$F

e. When mapping the heat transfer coefficient for a thermal analysis, make a selection in the **HTC Type** drop-down menu to determine how the heat transfer coefficient h_{eff} is calculated.

ref-temp
calculates h_{eff} using Equation 33.41 (p. 1995), where T_{ref} is the reference temperature defined in the Reference Values Task Page (p. 2502). Note that this option has the same definition as the field variable **Surface Heat Transfer Coef.**, as described in Alphabetical Listing of Field Variables and Their Definitions (p. 1957).

cell-temp
calculates h_{eff} using the general form of Equation 33.41 (p. 1995), but defines T_{ref} as the temperature of the cell adjacent to the face.

wall-func-htc
calculates h_{eff} using Equation 33.55 (p. 2001). Note that this option has the same definition as the field variable **Wall Func. Heat Tran. Coef.**, as described in Alphabetical Listing of Field Variables and Their Definitions (p. 1957).

f. Click **Write** to write an output file in which the ANSYS Fluent data has been mapped to the mesh of the input file.

The input file will be released from memory when the output file is written.

3.21. Saving Picture Files

Graphic window displays can be saved in various formats (including TIFF and PostScript). There can be slight differences between pictures and the displayed graphics windows, since pictures are generated using the internal software renderer, while the graphics windows may use specialized graphics hardware for optimum performance.

Many systems provide a utility to "dump" the contents of a graphics window into a raster file. This is generally the fastest method of generating a picture (since the scene is already rendered in the graphics window), and guarantees that the picture is identical to the window.

For additional information, see the following sections:
3.21.1. Using the Save Picture Dialog Box
3.21.2. Picture Options for PostScript Files

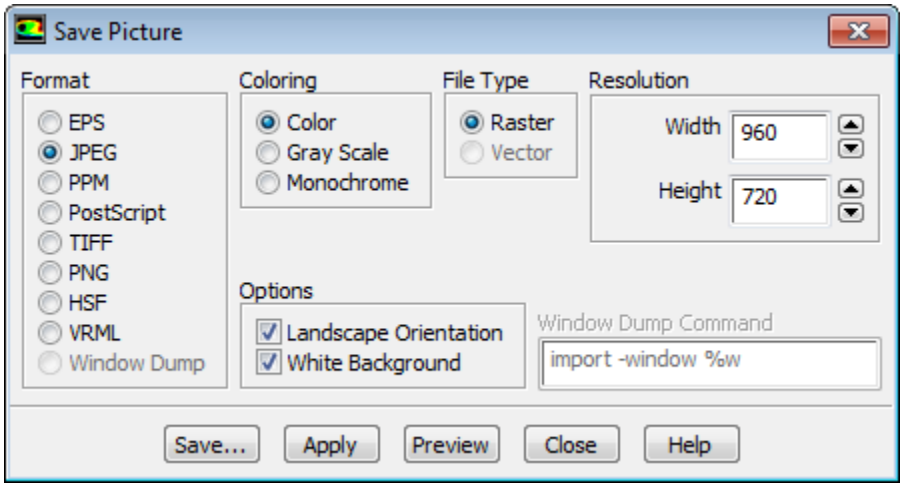
3.21.1. Using the Save Picture Dialog Box

To set picture parameters and save picture files, use the Save Picture Dialog Box (p. 2568) (Figure 3.15: The Save Picture Dialog Box (p. 103)).

File → **Save Picture...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.15: The Save Picture Dialog Box



For your convenience, this dialog box may also be opened using the **Save Picture** button ( ) in the standard toolbar.

The procedure for saving a picture file is as follows:

1. Choose the picture file **Format**.
2. Set the **Coloring**.
3. Specify the **File Type**, if applicable.
4. Define the **Resolution**, if applicable.
5. Set the appropriate **Options**.
6. If you are generating a window dump, specify the **Window Dump Command**.
7. (optional) Preview the result by clicking **Preview**.
8. Click the **Save...** button and enter the file name in the resulting **Select File** dialog box. See Automatic Numbering of Files (p. 43) for information on special features related to file name specification.

If you are not ready to save a picture but want to save the current picture settings, click the **Apply** button instead of the **Save...** button. The applied settings become the defaults for subsequent pictures.

3.21.1.1. Choosing the Picture File Format

To choose the picture file format, select one of the following items in the **Format** list:

EPS
(Encapsulated PostScript) output is the same as PostScript output, with the addition of Adobe Document Structuring Conventions (v2) statements. Currently, no preview bitmap is included in EPS output. Often, programs that import EPS files use the preview bitmap to display on-screen, although the actual vector PostScript information is used for printing (on a PostScript device). You can save EPS files in raster or vector format.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

JPEG
is a common raster file format.

PPM
output is a common raster file format.

PostScript
is a common vector file format. You can also choose to save a PostScript file in raster format.

TIFF
is a common raster file format.

PNG
is a common raster file format.

HSF
is HOOPS Visualize Stream Format, a highly-compressible and streamable 2D/3D file format.

VRML
is a graphics interchange format that allows export of 3D geometrical entities that you can display in the ANSYS Fluent graphics window. This format can commonly be used by VR systems and the 3D geometry can be viewed and manipulated in a web-browser graphics window.

Important

Non-geometric entities such as text, titles, color bars, and orientation axis are not exported. In addition, most display or visibility characteristics set in ANSYS Fluent, such as lighting, shading method, transparency, face and edge visibility, outer face culling, and hidden line removal, are not explicitly exported but are controlled by the software used to view the VRML file.

Window Dump
(Linux systems only) selects a window dump operation for generating the picture. With this format, you must specify the appropriate **Window Dump Command**.

Important

The advantage to saving the animation sequence using the **HSF File** option is that these files are highly compressible and can be viewed and interacted with using a "HOOPS Viewer" application on iOS and Android equipped devices. The "HOOPS Viewer" application can be downloaded to iOS and Android equipped devices from the "App Store" and "Google Play" respectively.

An advantage to saving the animation sequence using the **PPM Image** option is that you can use the separate pixmap image files for the creation of a single GIF file. GIF file creation can be done quickly with graphics tools provided by other third-party graphics packages such as ImageMagick, that is, `animate` or `convert`. For example, if you save the PPM files starting with the string `sequence-2`, and you are using the ImageMagick software, you can use the `convert` command with the `-adjoin` option to create a single GIF file out of the sequence using the following command.

```
convert -adjoin sequence-2_00*.ppm sequence2.gif
```

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.21.1.2. Specifying the Color Mode

For all formats except VRML, HSF, and the window dump, specify the type of **Coloring** you want to use for the picture file.

- Select **Color** for a color-scale copy.
- Select **Gray Scale** for a gray-scale copy.
- Select **Monochrome** for a black-and-white copy.

Most monochrome PostScript devices render **Color** images in shades of gray, but to ensure that the color ramp is rendered as a linearly-increasing gray ramp, you should select **Gray Scale**.

3.21.1.3. Choosing the File Type

When you save an EPS (Encapsulated PostScript) or PostScript file, choose one of the following under **File Type**:

- A **Raster** file defines the color of each individual pixel in the image. Raster files have a fixed resolution. The supported raster formats are EPS, JPEG, PPM, PostScript, TIFF, and PNG.
- A **Vector** file defines the graphics image as a combination of geometric primitives like lines, polygons, and text. Vector files are usually scalable to any resolution. The supported vector formats include EPS, PostScript, and VRML.

Important

For the quickest print time, you can save vector files for simple 2D displays and raster files for complicated scenes.

3.21.1.4. Defining the Resolution

For raster picture files (that is, JPEG, PPM, TIFF, and PNG), you can control the resolution of the picture image by specifying the size (in pixels). Set the desired **Width** and **Height** under **Resolution**. If the **Width** and **Height** are both zero, the picture is generated at the same resolution as the active graphics window. To check the size of the active window in pixels, click **Info** in the **Display Options** dialog box.

For EPS and PostScript files, specify the resolution in dots per inch (**DPI**) instead of setting the width and height.

3.21.1.5. Picture Options

For all picture formats except VRML, HSF, and the window dump, you can control two additional settings under **Options**:

- Specify the orientation of the picture using the **Landscape Orientation** button. If this option is turned on, the picture is made in landscape mode; otherwise, it is made in portrait mode.
- Control the foreground/background color using the **White Background** option. If this option is enabled, the picture is saved with a white background, and (if you are using the classic color scheme for the graphics window) the foreground color is changed to black.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.21.2. Picture Options for PostScript Files

ANSYS Fluent provides options that allow you to save PostScript files that can be printed more quickly. The following options are found in the `display/set/picture/driver/ post-format` text menu:

`fast-raster`
enables a raster file that may be larger than the standard raster file, but will print much more quickly.

`raster`
enables the standard raster file.

`rle-raster`
enables a run-length encoded raster file that is about the same size as the standard raster file, but will print slightly more quickly. This is the default file type.

`vector`
enables the standard vector file.

3.21.2.1. Window Dumps (Linux Systems Only)

If you select the **Window Dump** format, the program uses the specified **Window Dump Command** to save the picture file. For example, if you want to use `xwd` to capture a window, set the **Window Dump Command** to

```
xwd -id %w >
```

When the dump occurs, ANSYS Fluent automatically interprets `%w` to be the ID number of the active window.

When you click the **Save...** button, the **Select File** dialog box appears. Enter the file name for the output from the window dump (for example, `myfile.xwd`).

If you are planning to make an animation, save the window dumps into numbered files, using the %n variable. To do this, use the **Window Dump Command** (`xwd -id %w`), but for the file name in the **Select File** dialog box enter `myfile%n.xwd`. Each time a new window dump is created, the value of %n increases by one. So there is no need to tack numbers onto the picture file names manually.

To use the **ImageMagick** animate program, saving the files in MIFF format (the native **ImageMagick** format) is more efficient. In such cases, use the **ImageMagick** tool `import`. Set the default **Window Dump Command** enter

```
import -window %w
```

Click **Save...** to open the **Select File** dialog box. Specify the output format to be MIFF by using the `.miff` suffix at the end of file name.

The window dump feature is both, system and graphics-driver specific. Thus the commands available for dumping windows depends on the particular configuration.

The window dump captures the window exactly as it is displayed, including resolution, colors, transparency, and so on. For this reason, all of the inputs that control these characteristics are disabled in the **Save Picture** dialog box when you enable the **Window Dump** format. If you are using an 8-bit graphics display, use one of the built-in raster drivers (for example, TIFF) to generate higher-quality 24-bit color output rather than dumping the 8-bit window.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.21.2.2. Previewing the Picture Image

Before saving a picture file, you have the option of previewing what the saved image will look like. Click **Preview** to open a new window that will display the graphics using the current settings. This allows you to investigate the effects of different options interactively before saving the final, approved picture.

3.22. Setting Data File Quantities

By default, the information saved in a data file includes a standard set of quantities that were computed during the calculation. These quantities are specifically suitable for postprocessing and restarting solutions in ANSYS Fluent. If, however, you plan to postprocess the data file in an application other than ANSYS Fluent (such as ANSYS CFD-Post) you may want to include additional quantities that are derived from the standard quantities.

Note that some standard quantities are also listed as additional quantities. If using ANSYS CFD-Post, where a standard quantity has a corresponding entry in the additional quantity list, the latter should be selected. This is because ANSYS CFD-Post requires that some standard quantities be derived into a specific form.

Note

Not all standard quantities will be available in CFD-Post for postprocessing. An example of such a quantity is the Mach number.

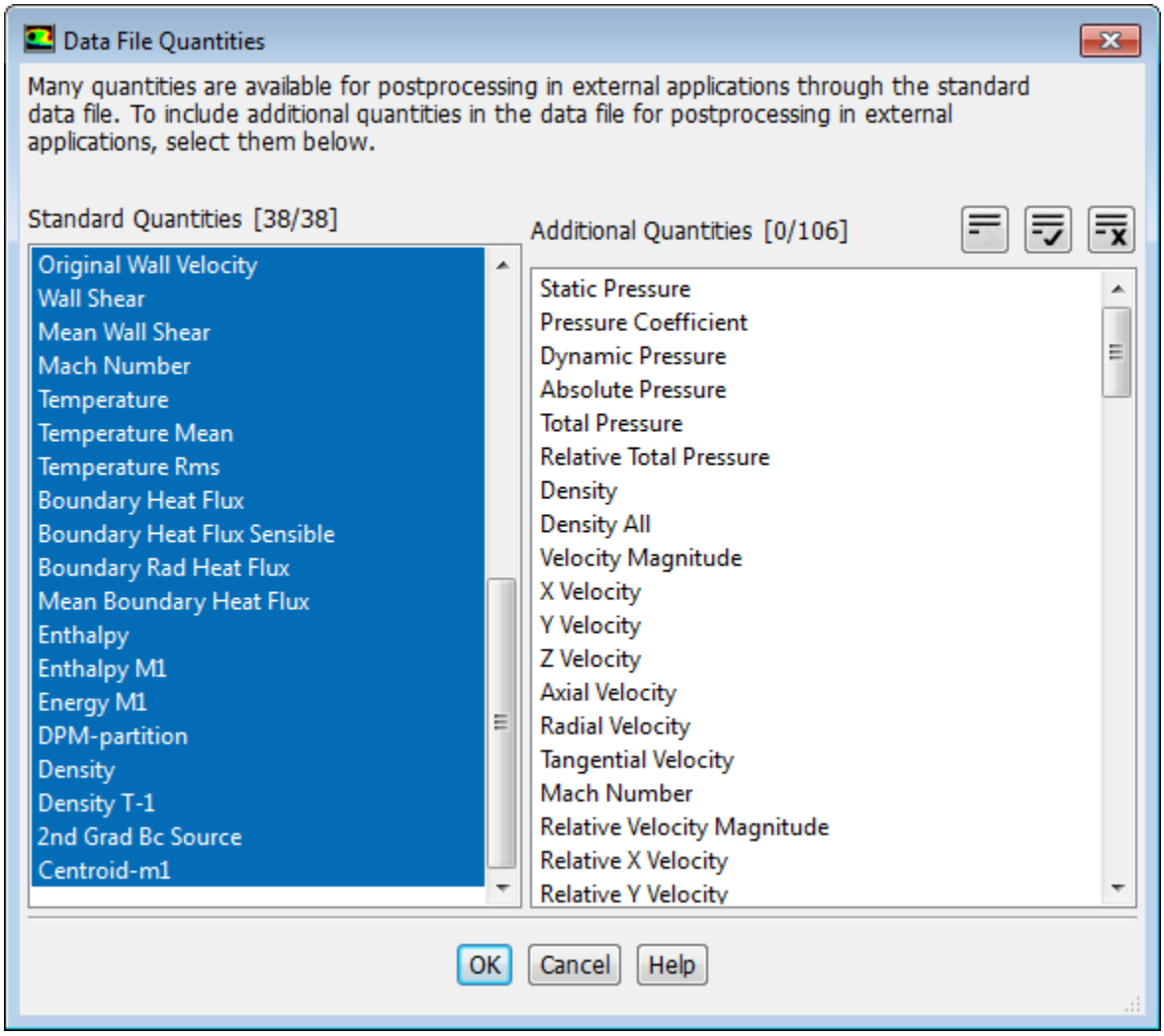
The procedure for generating a data file with additional quantities is as follows:

1. Set up the case file for your simulation.
2. Initialize the solution using the Solution Initialization Task Page (p. 2520).
3. Specify the quantities to be written in the data file, using the **Data File Quantities** dialog box (Figure 3.16: The Data File Quantities Dialog Box (p. 108)).

   File → **Data File Quantities...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.16: The Data File Quantities Dialog Box



a. View the **Standard Quantities** list to see what will be saved in the data file by default. Note that you cannot deselect any of the **Standard Quantities**.

b. Select the additional quantities you want saved from the **Additional Quantities** selection list.

c. Click **OK**.

4. Save the case file. Note that the data file quantities you specified in the previous step will be saved as part of the case file.

5. Run the calculation and save the data file. This can be done as separate steps, or as one step if you have selected the automatic saving of data files via the Calculation Activities Task Page (p. 2525).

The **Data File Quantities** dialog box can also be opened by clicking the **Data File Quantities...** button in the **Autosave Case/Data** dialog box.

3.23. The .fluent File

When starting up, ANSYS Fluent looks in your home folder for an optional file called `.fluent`. If it finds the file, it loads it with the Scheme `load` function. This file can contain Scheme functions that customize the code's operation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The `.fluent` file can also contain TUI commands that are executed via the Scheme function `ti-menu-load-string`. For example, if the `.fluent` file contains

```
(ti-menu-load-string "file read-case test.cas")
```

then ANSYS Fluent will read in the case file `test.cas`. For more details about the function `ti-menu-load-string`, see Text Menu Input from Character Strings (p. 35).

Important

Another optional file, `.tgrid`, if present, is also loaded at start up. This file may contain Scheme functions that customize the operation of the code in meshing mode. When both the `.fluent` and `.tgrid` files are present, the `.tgrid` file will be loaded first, followed by the `.fluent` file, when the solution mode is launched. Hence, the functions in the `.fluent` file will take precedence over those in the `.tgrid` file for the solution mode.

The `.fluent` file is not loaded automatically when switching to solution mode from meshing mode. You will need to load the file separately using the Scheme load function, if needed.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 4: Unit Systems

This chapter describes the units used in ANSYS Fluent and how you can control them. Information is organized into the following sections:

4.1. Restrictions on Units
4.2. Units in Mesh Files
4.3. Built-In Unit Systems in ANSYS Fluent
4.4. Customizing Units

ANSYS Fluent enables you to work in any unit system, including inconsistent units. Thus, for example, you may work in British units with heat input in Watts or you may work in SI units with length defined in inches. This is accomplished by providing ANSYS Fluent with a correct set of conversion factors between the units you want to use and the standard SI unit system that is used internally by the solver. ANSYS Fluent uses these conversion factors for input and output, internally storing all parameters and performing all calculations in SI units. Both solvers always prompt you for the units required for all dimensional inputs.

Units can be altered part-way through a problem setup and/or after you have completed your calculation. If you have entered some parameters in SI units and then you switch to British, all of your previous inputs (and the default prompts) are converted to the new unit system. If you have completed a simulation in SI units but you would like to report the results in any other units, you can alter the unit system and ANSYS Fluent will convert all of the problem data to the new unit system when results are displayed. As noted above, all problem inputs and results are stored in SI units internally. This means that the parameters stored in the case and data files are in SI units. ANSYS Fluent simply converts these values to your unit system at the interface level.

4.1. Restrictions on Units

It is important to note that the units for some inputs in ANSYS Fluent are different from the units used for the rest of the problem setup.

- You must always define the following in SI units, regardless of the unit system you are using:
  - Boundary profiles (see Profiles (p. 435))
  - Source terms (see Defining Mass, Momentum, Energy, and Other Sources (p. 297))
  - Custom field functions (see Custom Field Functions (p. 2003))
  - Data in externally-created XY plot files (see XY Plots of File Data (p. 1844))
  - User-defined functions (See the Fluent Customization Manual for details about user-defined functions.)
- If you define a material property by specifying a temperature-dependent polynomial or piecewise-polynomial function, remember that temperature in the function is always in units of Kelvin or Rankine. If you are using Celsius or Kelvin as your temperature unit, then polynomial coefficient values must be entered in terms of Kelvin; if you are using Fahrenheit or Rankine as the temperature unit, values must be entered in

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

terms of Rankine. See Defining Properties Using Temperature-Dependent Functions (p. 475) for information about temperature-dependent material properties.

4.2. Units in Mesh Files

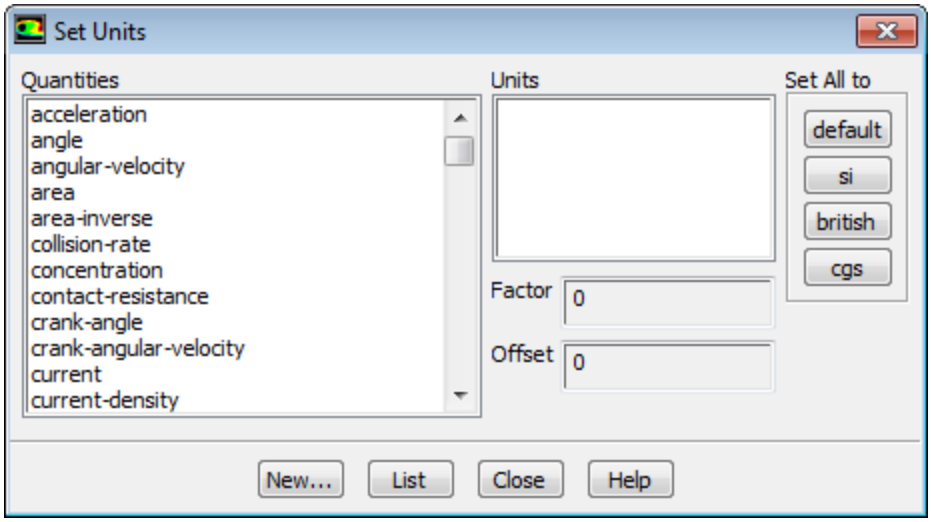
Some mesh generators allow you to define a set of units for the mesh dimensions. However, when you read the mesh into ANSYS Fluent, it is always assumed that the unit of length is meters. If this is not true, you must scale the mesh, as described in Scaling the Mesh (p. 226).

4.3. Built-In Unit Systems in ANSYS Fluent

ANSYS Fluent provides four built-in unit systems: British, SI, CGS, and "default", all of which can be selected in the **Set Units** dialog box (Figure 4.1: The Set Units Dialog Box (p. 112)), using the buttons under the **Set All to** heading. To display the **Set Units** dialog box, right-click **General** and click **Units...**, under **Setup** in the tree.

Setup → General → Units

Figure 4.1: The Set Units Dialog Box



To choose the English Engineering standard for all units, click the **british** button; to select the International System of units (SI) standard for all units, click the **si** button; to choose the CGS (centimeter-gram-second) standard for all units, click the **cgs** button; and to return to the "default" system, click the **default** button. The default system of units is similar to the SI system, but uses degrees instead of radians for angles. Clicking on one of the buttons under **Set All to** will immediately change the unit system. You can then close the dialog box if you are not interested in customizing any units.

Changing the unit system in the **Set Units** dialog box causes all future inputs that have units to be based on the newly selected unit system.

4.4. Customizing Units

If you would like a mixed unit system, or any unit system different from the four supplied by ANSYS Fluent (and described in Built-In Unit Systems in ANSYS Fluent (p. 112)), you can use the **Set Units** dialog box (Figure 4.1: The Set Units Dialog Box (p. 112)) to select an available unit or specify your own unit name and conversion factor for each quantity.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For additional information, see the following sections:
4.4.1. Listing Current Units
4.4.2. Changing the Units for a Quantity
4.4.3. Defining a New Unit

4.4.1. Listing Current Units

Before customizing units for one or more quantities, you may want to list the current units. You can do this by clicking the **List** button at the bottom of the **Set Units** dialog box. ANSYS Fluent will print out a list (in the text window) containing all quantities and their current units, conversion factors, and offsets.

4.4.2. Changing the Units for a Quantity

ANSYS Fluent will enable you to modify the units for individual quantities. This is useful for problems in which you want to use one of the built-in unit systems, but you want to change the units for one quantity (or for a few). For example, you may want to use SI units for your problem, but the dimensions of the geometry are given in inches. You can select the SI unit system, and then change the unit of length from meters to inches.

To change the units for a particular quantity, you will follow these two steps:

1. Select the quantity in the **Quantities** list (they are arranged in alphabetical order).

2. Choose a new unit from those that are available in the **Units** list.

For the example cited above, you would choose **length** in the **Quantities** list, and then select **in** in the **Units** list. The **Factor** will automatically be updated to show 0.0254 meters/inch. (See Figure 4.1: The Set Units Dialog Box (p. 112).) If there was a nonzero offset for the new unit, the **Offset** field would also be updated. For example, if you were using SI units but wanted to define temperature in Celsius instead of Kelvin, you would select **temperature** in the **Quantities** list and **c** in the **Units** list. The **Factor** would change to 1, and the **Offset** would change to 273.15. Once you have selected the quantity and the new unit, no further action is needed, unless you want to change the units for another quantity by following the same procedure.

4.4.3. Defining a New Unit

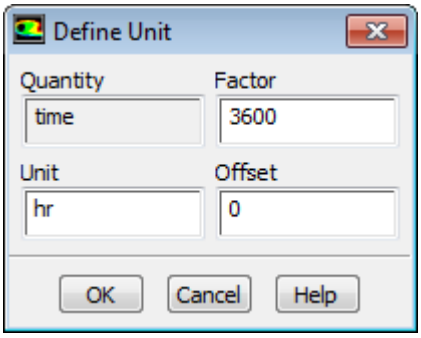
To create a new unit to be used for a particular quantity, you will follow the procedure below:

1. In the **Set Units** dialog box, select the quantity in the **Quantities** list.

2. Click the **New...** button and the **Define Unit** dialog box (Figure 4.2: The Define Unit Dialog Box (p. 114)) will open. In this dialog box, the selected quantity will be shown in the **Quantity** field.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4.2: The Define Unit Dialog Box



3. Enter the name of your new unit in the **Unit** field, the conversion factor in the **Factor** field, and the offset in the **Offset** field.

4. Click **OK** in the **Define Unit** dialog box, and the new unit will appear in the **Set Units** dialog box.

For example, if you want to use hours as the unit of time, select **time** in the **Quantities** list in the **Set Units** dialog box and click the **New...** button. In the resulting **Define Unit** dialog box, enter `hr` for the **Unit** and `3600` for the **Factor**, as in Figure 4.2: The Define Unit Dialog Box (p. 114). Then click **OK**. The new unit **hr** will appear in the **Units** list in the **Set Units** dialog box, and it will be selected.

4.4.3.1. Determining the Conversion Factor

The conversion factor you specify (**Factor** in the **Define Unit** dialog box) tells ANSYS Fluent the number to multiply by to obtain the SI unit value from your customized unit value. Thus the conversion factor should have the form SI units/custom units. For example, if you want the unit of length to be inches, you should enter a conversion factor of 0.0254 meters/inch. If you want the unit of velocity to be feet/min, you can determine the conversion factor by using the following equation:

$$x\frac{\text{ft}}{\text{min}}\times\frac{0.3048\text{m}}{\text{ft}}\times\frac{\text{min}}{60\text{s}}=y\frac{\text{m}}{\text{s}} \tag{4.1}$$

You should enter a conversion factor of 0.0051, which is equal to 0.3048/60.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 5: Reading and Manipulating Meshes

ANSYS Fluent can import different types of meshes from various sources. You can modify the mesh by translating or scaling node coordinates, partitioning the domain for parallel processing, and merging or separating zones. You can convert all 3D meshes to polyhedral cells, except for pure hex meshes. Hexahedral cells are preserved during conversion. You can also obtain diagnostic information on the mesh, including memory usage and simplex, topological, and domain information. You can find out the number of nodes, faces, and cells in the mesh, determine the minimum and maximum cell volumes in the domain, and check for the proper numbers of nodes and faces per cell. These and other capabilities are described in the following sections.

5.1. Mesh Topologies
5.2. Mesh Requirements and Considerations
5.3. Mesh Sources
5.4. Non-Conformal Meshes
5.5. Overset Meshes
5.6. Checking the Mesh
5.7. Reporting Mesh Statistics
5.8. Converting the Mesh to a Polyhedral Mesh
5.9. Modifying the Mesh

See Adapting the Mesh (p. 1681) for information about adapting the mesh based on solution data and related functions, and Mesh Partitioning and Load Balancing (p. 2030) for details on partitioning the mesh for parallel processing.

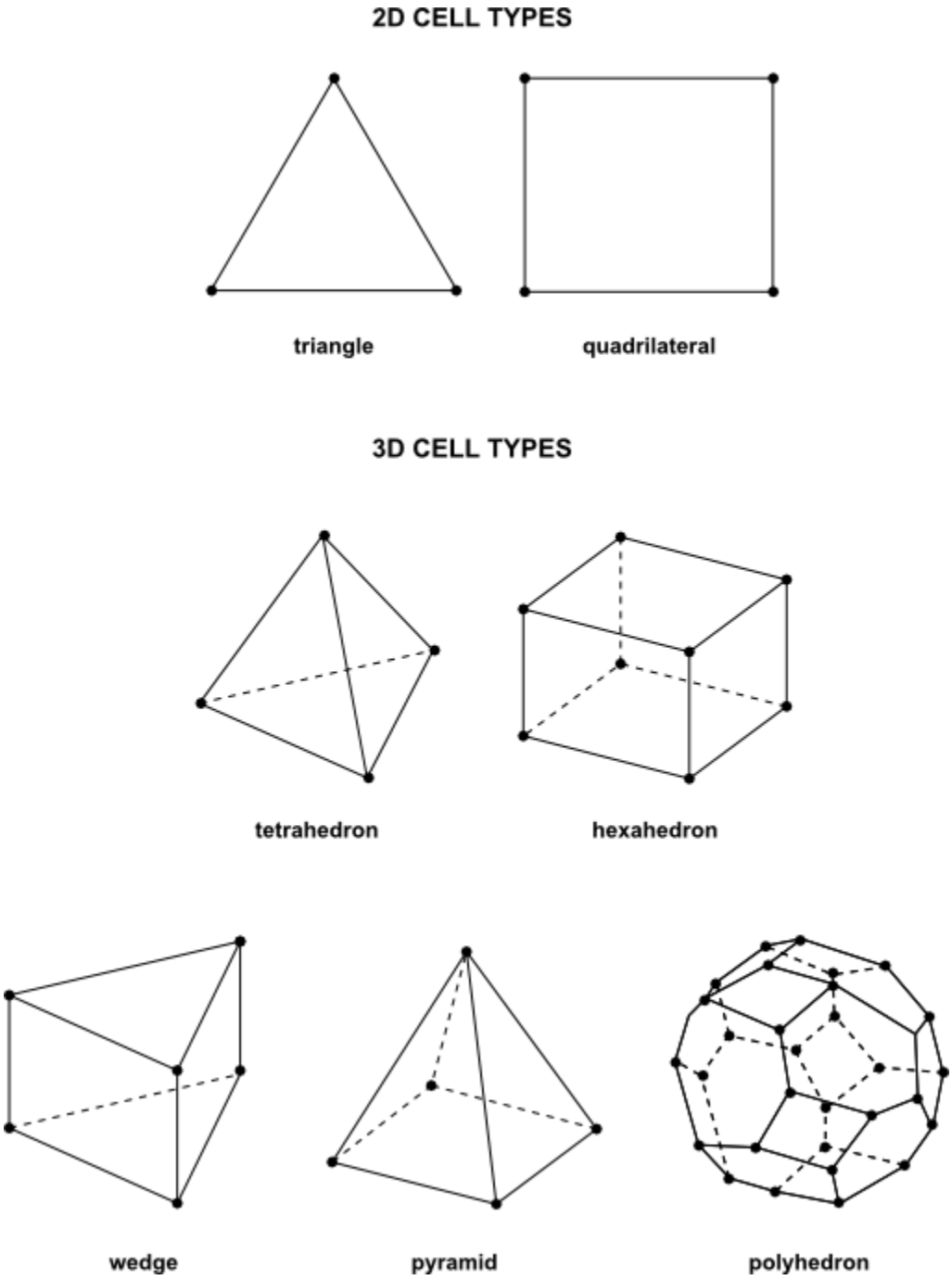
5.1. Mesh Topologies

As an unstructured solver, ANSYS Fluent uses internal data structures to assign an order to the cells, faces, and grid points in a mesh and to maintain contact between adjacent cells. Therefore, it does not require i,j,k indexing to locate neighboring cells. This gives you the flexibility to use the best mesh topology for your problem, as the solver does not force an overall structure or topology on the mesh.

For 2D meshes, quadrilateral and triangular cells are accepted, and for 3D meshes, hexahedral, tetrahedral, pyramid, wedge, and polyhedral cells can be used. Figure 5.1: Cell Types (p. 116) depicts each of these cell types. Both single-block and multi-block structured meshes, as well as hybrid meshes containing quadrilateral and triangular cells or hexahedral, tetrahedral, pyramid, and wedge cells are acceptable. ANSYS Fluent also accepts meshes with hanging nodes (that is, nodes on edges and faces that are not vertices of all the cells sharing those edges or faces) and hanging edges (that is, edges on faces that do not act as edges for both of the cells sharing those faces), although you may need to remove the hanging nodes/edges from interior walls. See Hanging Node Adaption in the Theory Guide for information about acceptable hanging nodes, and Using the `tpoly` Filter to Remove Hanging Nodes/Edges (p. 143) and Converting Cells with Hanging Nodes / Edges to Polyhedra (p. 205) for details on removing hanging nodes/edges. Meshes with non-conformal boundaries (that is, meshes with multiple subdomains in which the mesh node locations at the internal subdomain boundaries are not identical) are also acceptable. For details, see Non-Conformal Meshes (p. 150).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.1: Cell Types



Some examples of meshes that are valid for ANSYS Fluent are presented in Examples of Acceptable Mesh Topologies (p. 116). Different cell shapes and their face-node connectivity are explained in Face-Node Connectivity in ANSYS Fluent (p. 121). Choosing the Appropriate Mesh Type (p. 128) explains how to choose the mesh type that is best suited for your problem.

5.1.1. Examples of Acceptable Mesh Topologies

ANSYS Fluent can solve problems on a wide variety of meshes. Figure 5.2: Structured Quadrilateral Mesh for an Airfoil (p. 117)-Figure 5.13: Polyhedral Mesh (p. 121) show examples of meshes that are valid for ANSYS Fluent.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

O-type meshes, meshes with zero-thickness walls, C-type meshes, conformal block-structured meshes, multiblock structured meshes, non-conformal meshes, and unstructured triangular, tetrahedral, quadrilateral, hexahedral, and polyhedral meshes are all acceptable.

Note

Though ANSYS Fluent does not require a cyclic branch cut in an O-type mesh, it will accept a mesh that contains one.

Figure 5.2: Structured Quadrilateral Mesh for an Airfoil

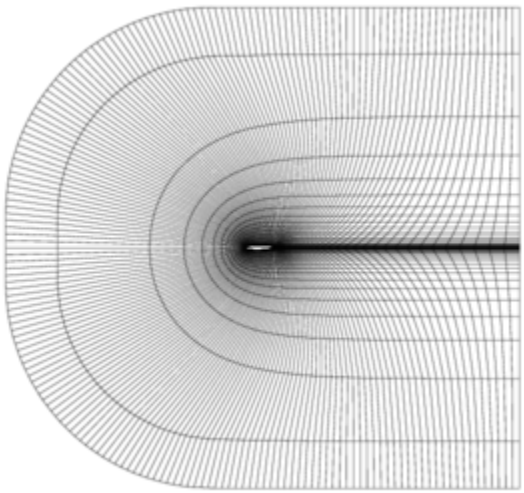
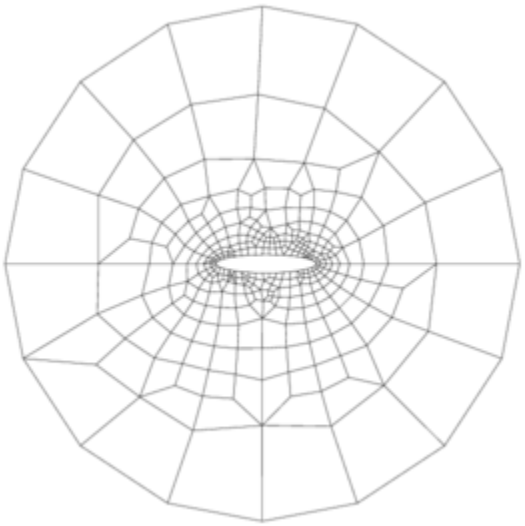


Figure 5.3: Unstructured Quadrilateral Mesh



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.4: Multiblock Structured Quadrilateral Mesh

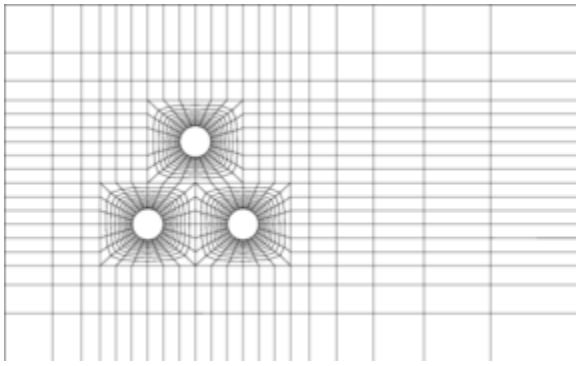


Figure 5.5: O-Type Structured Quadrilateral Mesh

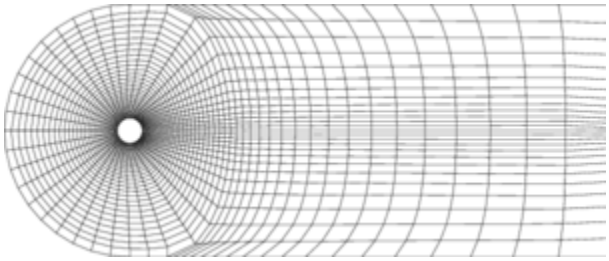
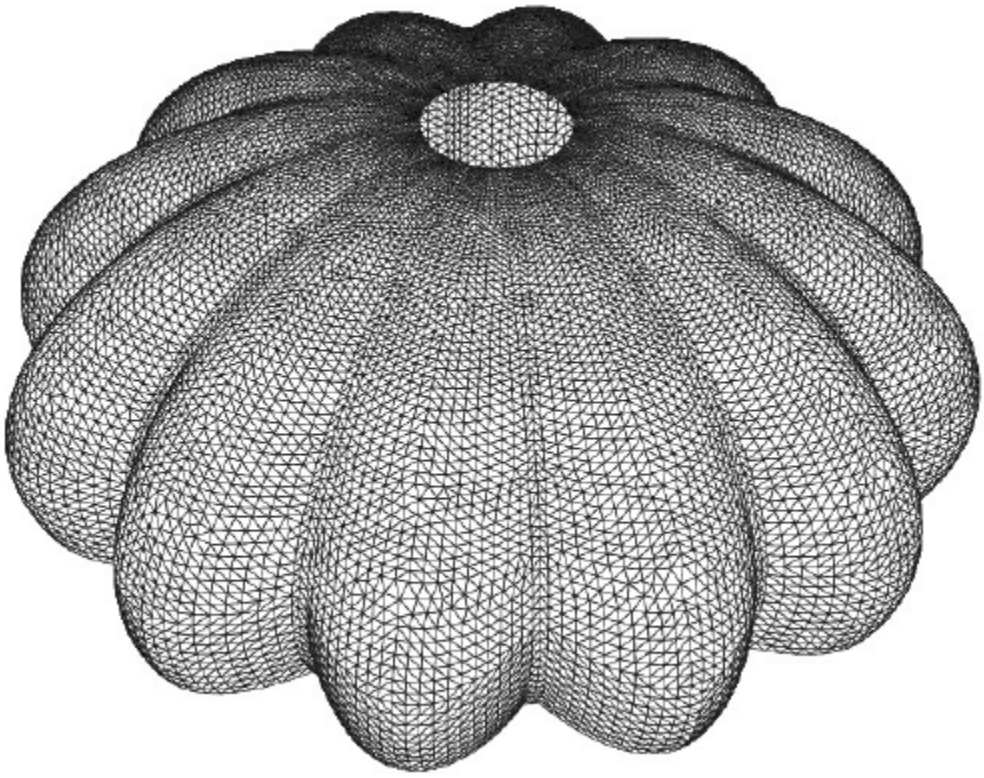


Figure 5.6: Parachute Modeled With Zero-Thickness Wall



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.7: C-Type Structured Quadrilateral Mesh

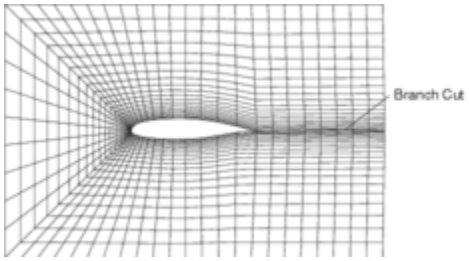


Figure 5.8: 3D Multiblock Structured Mesh

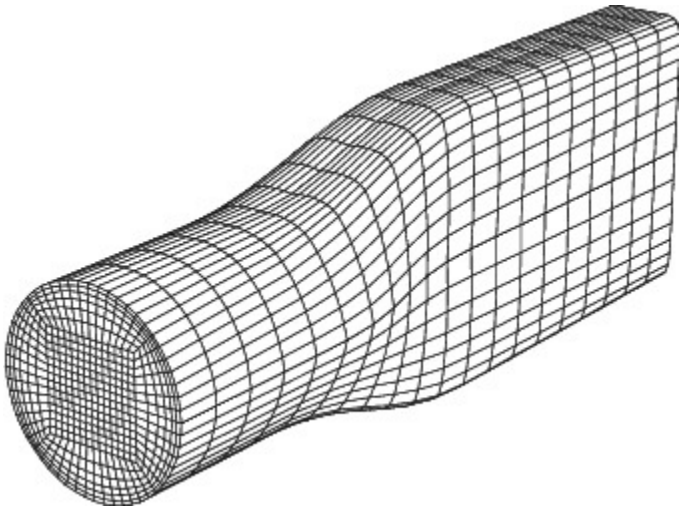
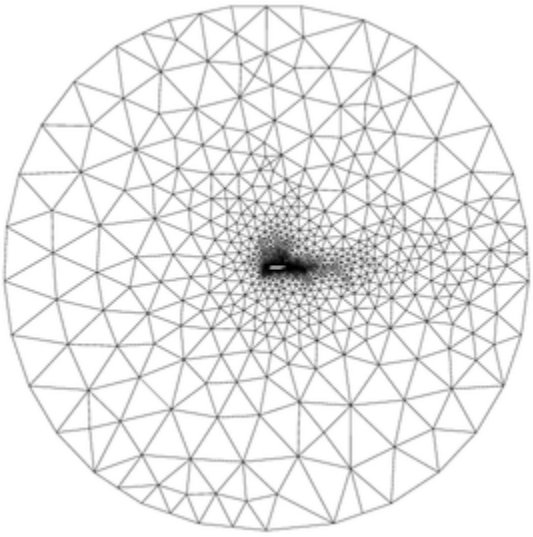


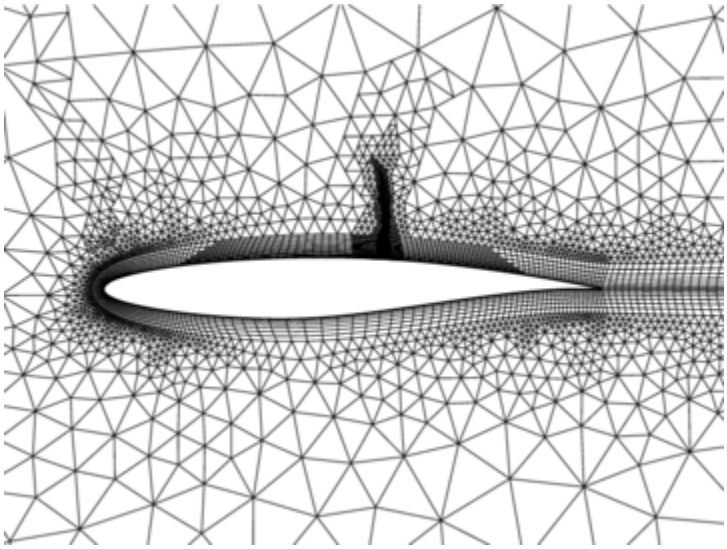
Figure 5.9: Unstructured Triangular Mesh for an Airfoil



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.10: Unstructured Tetrahedral Mesh

Figure 5.11: Hybrid Triangular/Quadrilateral Mesh with Hanging Nodes



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.12: Non-Conformal Hybrid Mesh for a Rotor-Stator Geometry

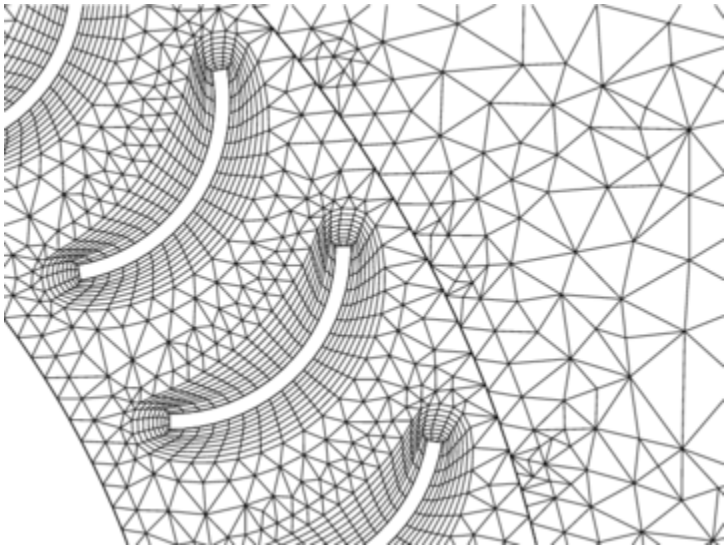


Figure 5.13: Polyhedral Mesh

5.1.2. Face-Node Connectivity in ANSYS Fluent

This section contains information about the connectivity of faces and their related nodes in terms of node number and face number.

Face-node connectivity for the following cell shapes is explained here:

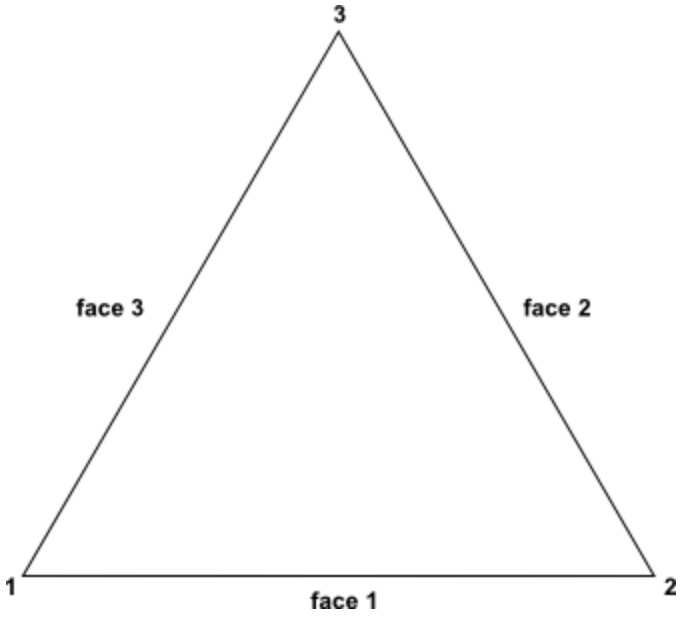
- triangular (Figure 5.14: Face and Node Numbering for Triangular Cells (p. 122))
- quadrilateral (Figure 5.15: Face and Node Numbering for Quadrilateral Cells (p. 123))
- tetrahedral (Figure 5.16: Face and Node Numbering for Tetrahedral Cells (p. 124))
- wedge (Figure 5.17: Face and Node Numbering for Wedge Cells (p. 125))
- pyramidal (Figure 5.18: Face and Node Numbering for Pyramidal Cells (p. 126))
- hex (Figure 5.19: Face and Node Numbering for Hex Cells (p. 127))
- polyhedral (Figure 5.20: An Example of a Polyhedral Cell (p. 128))

This information is useful in interfacing with ANSYS Fluent.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.1.2.1. Face-Node Connectivity for Triangular Cells

Figure 5.14: Face and Node Numbering for Triangular Cells

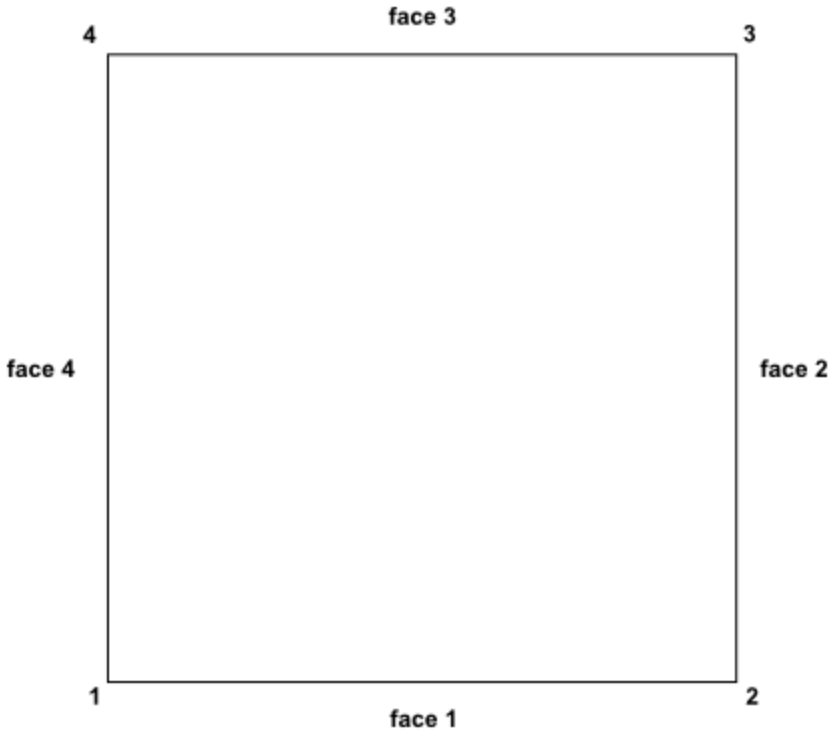


Face	Associated Nodes
Face 1	1-2
Face 2	2-3
Face 3	3-1

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.1.2.2. Face-Node Connectivity for Quadrilateral Cells

Figure 5.15: Face and Node Numbering for Quadrilateral Cells

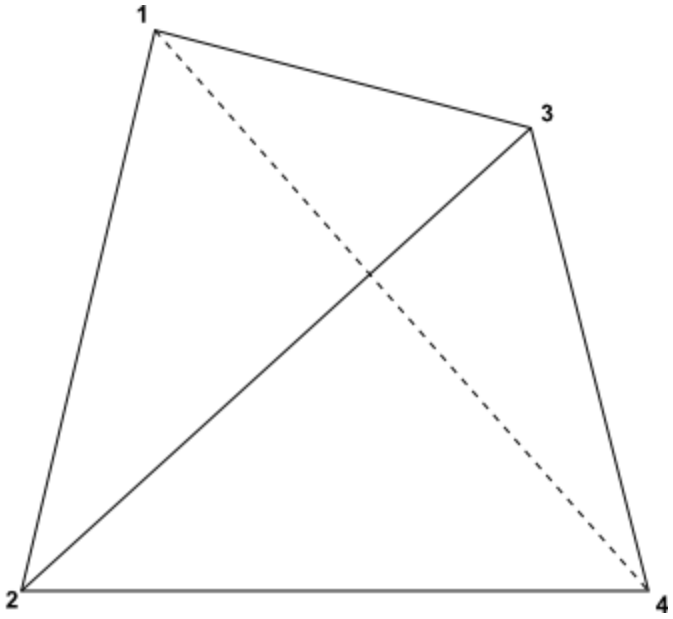


Face	Associated Nodes
Face 1	1-2
Face 2	2-3
Face 3	3-4
Face 4	4-1

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.1.2.3. Face-Node Connectivity for Tetrahedral Cells

Figure 5.16: Face and Node Numbering for Tetrahedral Cells

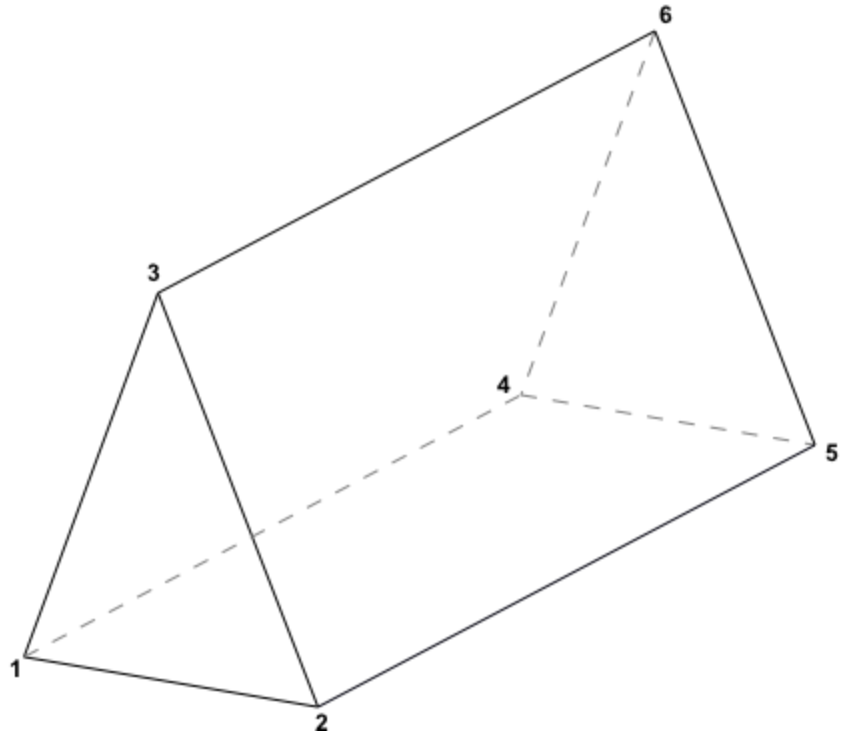


Face	Associated Nodes
Face 1	4-3-2
Face 2	3-4-1
Face 3	2-1-4
Face 4	3-1-2

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.1.2.4. Face-Node Connectivity for Wedge Cells

Figure 5.17: Face and Node Numbering for Wedge Cells

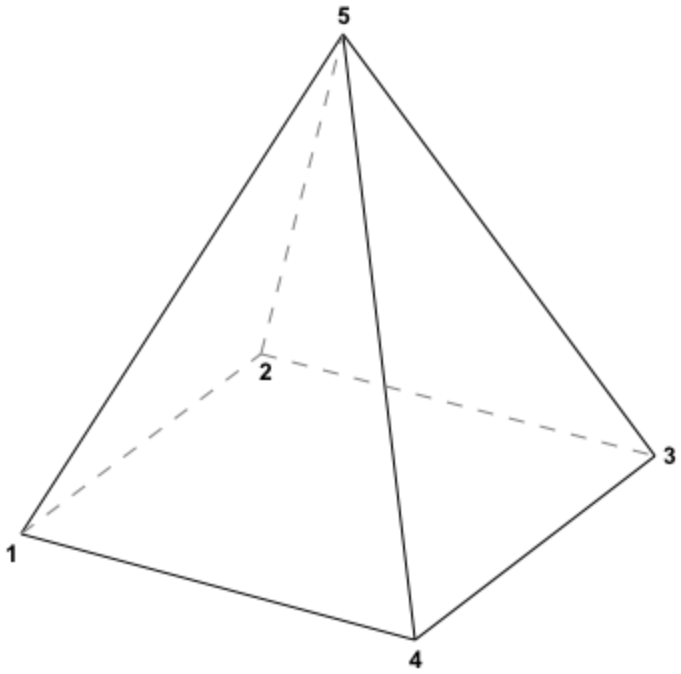


Face	Associated Nodes
Face 1	1-2-3
Face 2	1-4-5-2
Face 3	2-5-6-3
Face 4	3-6-4-1
Face 5	4-6-5

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.1.2.5. Face-Node Connectivity for Pyramidal Cells

Figure 5.18: Face and Node Numbering for Pyramidal Cells

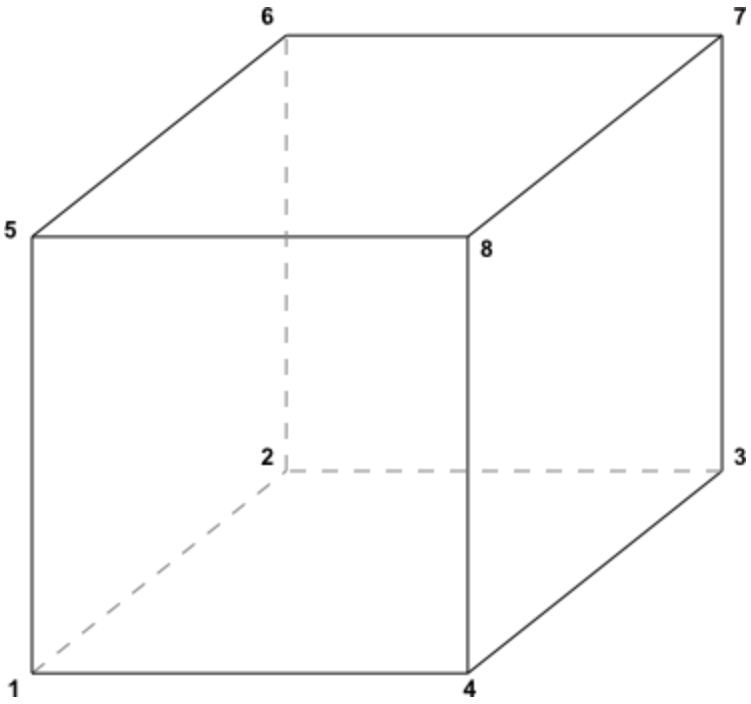


Face	Associated Nodes
Face 1	1-2-3-4
Face 2	1-5-2
Face 3	2-5-3
Face 4	3-5-4
Face 5	4-5-1

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.1.2.6. Face-Node Connectivity for Hex Cells

Figure 5.19: Face and Node Numbering for Hex Cells

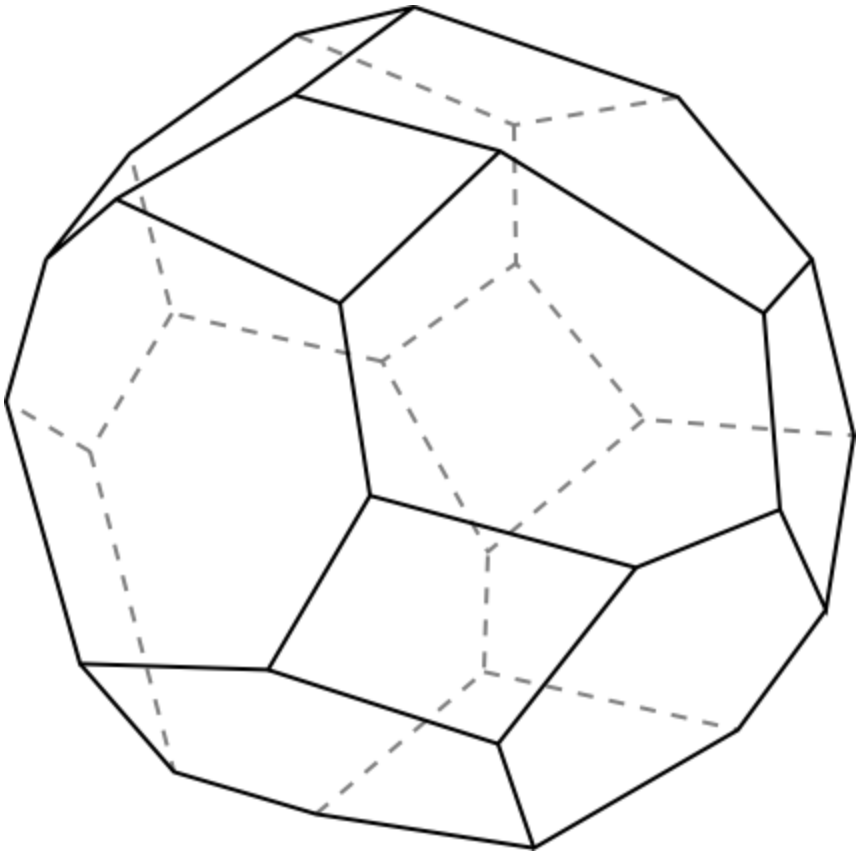


Face	Associated Nodes
Face 1	1-2-3-4
Face 2	1-5-6-2
Face 3	2-6-7-3
Face 4	4-3-7-8
Face 5	1-4-8-5
Face 6	5-8-7-6

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.1.2.7. Face-Node Connectivity for Polyhedral Cells

Figure 5.20: An Example of a Polyhedral Cell



For polyhedral cells, there is no explicit face and node numbering as with the other cell types.

5.1.3. Choosing the Appropriate Mesh Type

ANSYS Fluent can use meshes composed of triangular or quadrilateral cells (or a combination of the two) in 2D, and tetrahedral, hexahedral, polyhedral, pyramid, or wedge cells (or a combination of these) in 3D. The choice of which mesh type to use will depend on your application. When choosing mesh type, consider the following issues:

- setup time
- computational expense
- numerical diffusion

5.1.3.1. Setup Time

Many flow problems solved in engineering practice involve complex geometries. The creation of structured or block-structured meshes (consisting of quadrilateral or hexahedral elements) for such problems can be extremely time-consuming if not impossible. Therefore, setup time for complex geometries is the major motivation for using unstructured meshes employing triangular or tetrahedral cells. However, if your geometry is relatively simple, there may be no saving in setup time with either approach.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Other risks of using structured or block-structured meshes with complicated geometries include the oversimplification of the geometry, mesh quality issues, and a less efficient mesh distribution (for example, fine resolution in areas of less importance) that results in a high cell count.

If you already have a mesh created for a structured code, it will save you time to use this mesh in ANSYS Fluent rather than regenerate it. This can be a motivation for using quadrilateral or hexahedral cells in your ANSYS Fluent simulation.

Note

ANSYS Fluent has a range of filters that allow you to import structured meshes from other codes, including FLUENT 4. For details, see Mesh Sources (p. 135).

5.1.3.2. Computational Expense

When geometries are complex or the range of length scales of the flow is large, a triangular/tetrahedral mesh can be created with far fewer cells than the equivalent mesh consisting of quadrilateral/hexahedral elements. This is because a triangular/tetrahedral mesh allows clustering of cells in selected regions of the flow domain. Structured quadrilateral/hexahedral meshes will generally force cells to be placed in regions where they are not needed. Unstructured quadrilateral/hexahedral meshes offer many of the advantages of triangular/tetrahedral meshes for moderately-complex geometries.

A characteristic of quadrilateral/hexahedral elements that might make them more economical in some situations is that they permit a much larger aspect ratio than triangular/tetrahedral cells. A large aspect ratio in a triangular/tetrahedral cell will invariably affect the skewness of the cell, which is undesirable as it may impede accuracy and convergence. Therefore, if you have a relatively simple geometry in which the flow conforms well to the shape of the geometry, such as a long thin duct, use a mesh of high-aspect-ratio quadrilateral/hexahedral cells. The mesh is likely to have far fewer cells than if you use triangular/tetrahedral cells.

Converting the entire domain of your (tetrahedral) mesh to a polyhedral mesh will result in a lower cell count than your original mesh. Although the result is a coarser mesh, convergence will generally be faster, possibly saving you some computational expense.

In summary, the following practices are generally recommended:

- For simple geometries, use quadrilateral/hexahedral meshes.
- For moderately complex geometries, use unstructured quadrilateral/hexahedral meshes.
- For relatively complex geometries, use triangular/tetrahedral meshes with wedge elements in the boundary layers.
- For extremely complex geometries, use pure triangular/tetrahedral meshes.

5.1.3.3. Numerical Diffusion

A dominant source of error in multidimensional situations is numerical diffusion (false diffusion). The term *false diffusion* is used because the diffusion is not a real phenomenon, yet its effect on a flow calculation is analogous to that of increasing the real diffusion coefficient.

The following comments can be made about numerical diffusion:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Numerical diffusion is most noticeable when the real diffusion is small, that is, when the situation is convection-dominated.
- All practical numerical schemes for solving fluid flow contain a finite amount of numerical diffusion. This is because numerical diffusion arises from truncation errors that are a consequence of representing the fluid flow equations in discrete form.
- The second-order and the MUSCL discretization scheme used in ANSYS Fluent can help reduce the effects of numerical diffusion on the solution.
- The amount of numerical diffusion is inversely related to the resolution of the mesh. Therefore, one way of dealing with numerical diffusion is to refine the mesh.
- Numerical diffusion is minimized when the flow is aligned with the mesh.

  This is the most relevant to the choice of the mesh. If you use a triangular/tetrahedral mesh, the flow can *never* be aligned with the mesh. If you use a quadrilateral/hexahedral mesh, this situation might occur, but *not for complex flows*. It is only in a *simple* flow, such as the flow through a long duct, in which you can rely on a quadrilateral/hexahedral mesh to minimize numerical diffusion. In such situations, it is advantageous to use a quadrilateral/hexahedral mesh, since you will be able to get a better solution with fewer cells than if you were using a triangular/tetrahedral mesh.
- If you would like higher resolution for a gradient that is perpendicular to a wall, you can create prism layers with higher aspect ratios near the wall.

5.2. Mesh Requirements and Considerations

This section contains information about special geometry/mesh requirements and general comments on mesh quality.

5.2.1. Geometry/Mesh Requirements
5.2.2. Mesh Quality

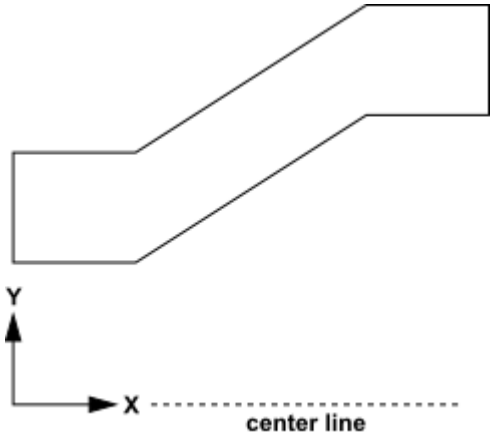
5.2.1. Geometry/Mesh Requirements

You should be aware of the following geometry setup and mesh construction requirements at the beginning of your problem setup:

- Axisymmetric geometries must be defined such that the axis of rotation is the x axis of the Cartesian coordinates used to define the geometry (Figure 5.21: Setup of Axisymmetric Geometries with the x Axis as the Centerline (p. 131)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.21: Setup of Axisymmetric Geometries with the x Axis as the Centerline



- ANSYS Fluent allows you to set up periodic boundaries using either conformal or non-conformal periodic zones. For conformal periodic boundaries, the periodic zones must have identical meshes.

  The conformal periodic boundaries can be created in the meshing mode of Fluent or GAMBIT when you are generating the volume mesh. See the Fluent Meshing User's Guide or the GAMBIT Modeling Guide for more information. Alternatively, you can create the conformal periodic boundaries in the solution mode of Fluent using the `mesh/modify-zones/make-periodic` text command. For details, see Creating Conformal Periodic Zones (p. 214).

  Although Fluent meshing mode and GAMBIT can produce true periodic boundaries, most CAD packages do not. If your mesh was created in such a package, set up a non-conformal interface with the periodic boundary condition option enabled in Fluent. For details, see The Periodic Boundary Condition Option (p. 153).

5.2.2. Mesh Quality

The quality of the mesh plays a significant role in the accuracy and stability of the numerical computation. Regardless of the type of mesh used in your domain, checking the quality of your mesh is essential. One important indicator of mesh quality that ANSYS Fluent allows you to check is a quantity referred to as the orthogonality. In order to determine the orthogonality of a given cell, the following quantities are calculated for each face i:

- the normalized dot product of the area vector of a face ($\vec{A}_i$) and a vector from the centroid of the cell to the centroid of that face ($\vec{f}_i$):

$$\frac{\vec{A}_i \cdot \vec{f}_i}{|\vec{A}_i||\vec{f}_i|} \tag{5.1}$$

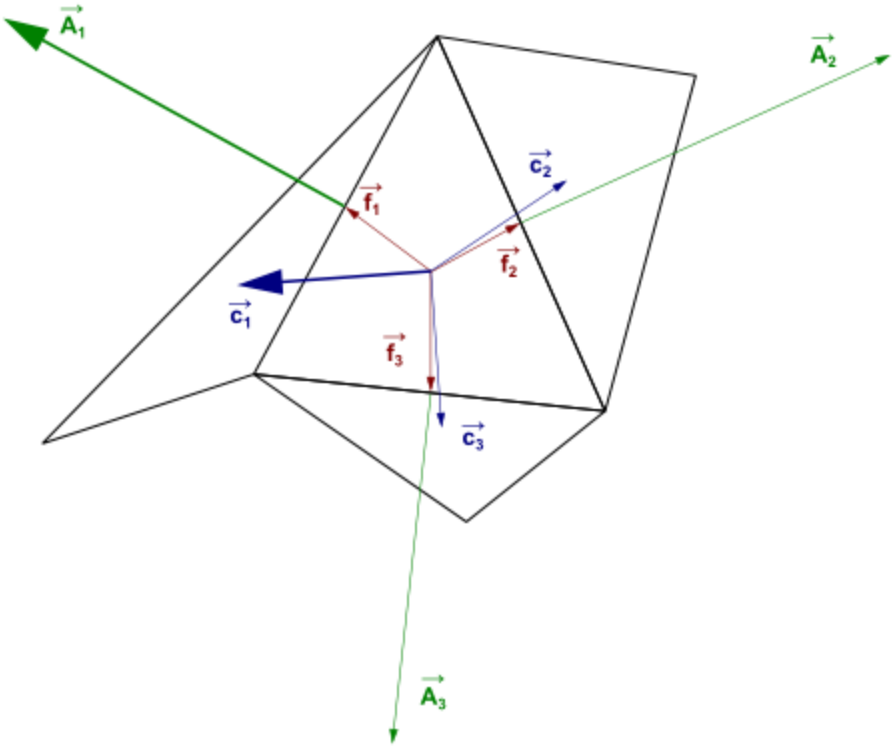
- the normalized dot product of the area vector of a face ($\vec{A}_i$) and a vector from the centroid of the cell to the centroid of the adjacent cell that shares that face ($\vec{c}_i$):

$$\frac{\vec{A}_i \cdot \vec{c}_i}{|\vec{A}_i||\vec{c}_i|} \tag{5.2}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The minimum value that results from calculating Equation 5.1 (p. 131) and Equation 5.2 (p. 131) for all of the faces is then defined as the orthogonality for the cell. Therefore, the worst cells will have an orthogonality closer to 0 and the best cells will have an orthogonality closer to 1. Figure 5.22: The Vectors Used to Compute Orthogonality (p. 132) illustrates the relevant vectors.

Figure 5.22: The Vectors Used to Compute Orthogonality



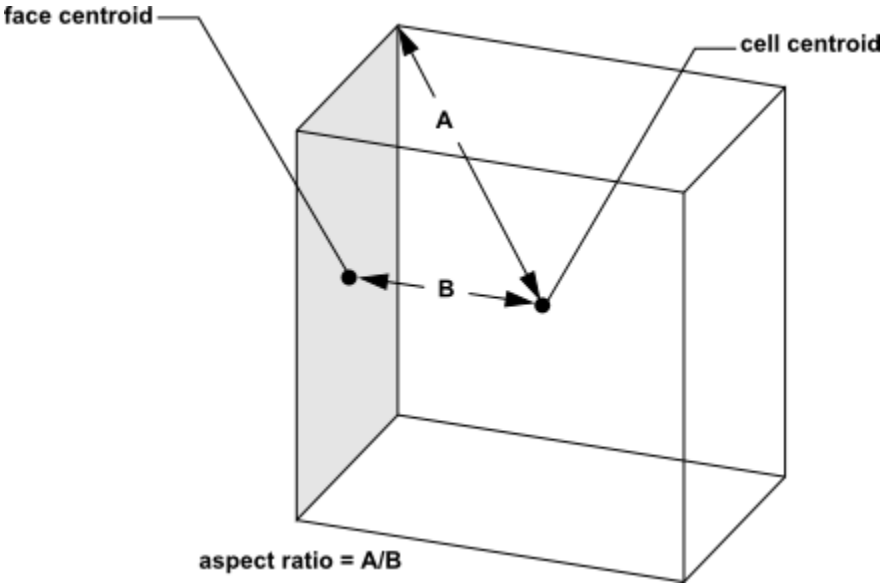
Orthogonal Quality depends on cell type:

- For tetrahedral, prism, and pyramid cells, the Orthogonal Quality is the minimum of the orthogonality and (1- cell skewness).
- For hexahedral and polyhedral cells, the Orthogonal Quality is the same as the orthogonality.

Another important indicator of the mesh quality is the aspect ratio. The aspect ratio is a measure of the stretching of a cell. It is computed as the ratio of the maximum value to the minimum value of any of the following distances: the normal distances between the cell centroid and face centroids (computed as a dot product of the distance vector and the face normal), and the distances between the cell centroid and nodes. For a unit cube (see Figure 5.23: Calculating the Aspect Ratio for a Unit Cube (p. 133)), the maximum distance is 0.866, and the minimum distance is 0.5, so the aspect ratio is 1.732. This type of definition can be applied on any type of mesh, including polyhedral.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.23: Calculating the Aspect Ratio for a Unit Cube



To check the quality of your mesh, you can use the **Report Quality** button in the **General** task page:

Setup → **General** → **Report Quality**

A message will be displayed in the console, such as the example that follows:

```
Mesh Quality:
 Orthogonal Quality ranges from 0 to 1, where values close to 0 correspond to low quality.
Minimum Orthogonal Quality = 6.07960e-01
Maximum Aspect Ratio = 5.42664e+00
```

If you would like more information about the quality displayed in the console (including additional quality metrics and the zones that have the cells with the lowest quality), set the `mesh/check-verbosity` text command to 2 (valid values are `0`, `1`, 2) prior to using the **Report Quality** button.

For information about how to improve poor quality cells, see Repairing Meshes (p. 194).

When evaluating whether the quality of your mesh is sufficient for the problem you are modeling, it is important to consider attributes such as mesh element distribution, cell shape, smoothness, and flow-field dependency. These attributes are described in the sections that follow.

5.2.2.1. Mesh Element Distribution

Since you are discretely defining a continuous domain, the degree to which the salient features of the flow (such as shear layers, separated regions, shock waves, boundary layers, and mixing zones) are resolved depends on the density and distribution of mesh elements. In many cases, poor resolution in critical regions can dramatically affect results. For example, the prediction of separation due to an adverse pressure gradient depends heavily on the resolution of the boundary layer upstream of the point of separation.

Resolution of the boundary layer (that is, mesh spacing near walls) also plays a significant role in the accuracy of the computed wall shear stress and heat transfer coefficient. This is particularly true in laminar flows where the mesh adjacent to the wall should obey

$$y_p\sqrt{\frac{u_\infty}{\nu x}} \leq 1 \tag{5.3}$$

where

y_p = distance to the wall from the adjacent cell centroid

u_∞ = free-stream velocity

ν = kinematic viscosity of the fluid

x = distance along the wall from the starting point of the boundary layer

Equation 5.3 (p. 133) is based upon the Blasius solution for laminar flow over a flat plate at zero incidence [100] (p. 2892).

Proper resolution of the mesh for turbulent flows is also very important. Due to the strong interaction of the mean flow and turbulence, the numerical results for turbulent flows tend to be more susceptible to mesh element distribution than those for laminar flows. In the near-wall region, different mesh resolutions are required depending on the near-wall model being used. See Model Hierarchy (p. 763) for guidelines.

In general, no flow passage should be represented by fewer than 5 cells. Most cases will require many more cells to adequately resolve the passage. In regions of large gradients, as in shear layers or mixing zones, the mesh should be fine enough to minimize the change in the flow variables from cell to cell. Unfortunately, it is very difficult to determine the locations of important flow features in advance. Moreover, the mesh resolution in most complicated 3D flow fields will be constrained by CPU time and computer resource limitations (that is, memory and disk space). Although accuracy increases with larger meshes, the CPU and memory requirements to compute the solution and postprocess the results also increase. Solution-adaptive mesh refinement can be used to increase and/or decrease mesh density based on the evolving flow field, and therefore provides the potential for more economical use of grid points (and hence reduced time and resource requirements). See Adapting the Mesh (p. 1681) for information on solution adaption.

5.2.2.2. Cell Quality

The quality of the cell (including its orthogonal quality, aspect ratio, and skewness) also has a significant impact on the accuracy of the numerical solution.

- *Orthogonal quality* is computed for cells using cell skewness and the vector from the cell centroid to each of its faces, the corresponding face area vector, and the vector from the cell centroid to the centroids of each of the adjacent cells (see Equation 5.1 (p. 131), Equation 5.2 (p. 131), and Figure 5.22: The Vectors Used to Compute Orthogonality (p. 132)). The worst cells will have an orthogonal quality closer to 0, with the best cells closer to 1. The minimum orthogonal quality for all types of cells should be more than 0.01, with an average value that is significantly higher.

- *Aspect ratio* is a measure of the stretching of the cell. As discussed in Computational Expense (p. 129), for highly anisotropic flows, extreme aspect ratios may yield accurate results with fewer cells. Generally, it is best to avoid sudden and large changes in cell aspect ratios in areas where the flow field exhibit large changes or strong gradients.

- *Skewness* is defined as the difference between the shape of the cell and the shape of an equilateral cell of equivalent volume. Highly skewed cells can decrease accuracy and destabilize the solution. For example, optimal quadrilateral meshes will have vertex angles close to 90 degrees, while triangular meshes should preferably have angles of close to 60 degrees and have all angles less than 90 degrees. A general rule is that the maximum skewness for a triangular/tetrahedral mesh in most flows should be kept below 0.95, with an average value that is significantly lower. A maximum value above 0.95 may lead to convergence difficulties and may require changing the solver controls, such as reducing under-relaxation factors and/or switching to the pressure-based coupled solver.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cell size change and face warp are additional quality measure that could affect stability and accuracy. See the Fluent Meshing User's Guide for more details.

5.2.2.3. Smoothness

Truncation error is the difference between the partial derivatives in the governing equations and their discrete approximations. Rapid changes in cell volume between adjacent cells translate into larger truncation errors. ANSYS Fluent provides the capability to improve the smoothness by refining the mesh based on the change in cell volume or the gradient of cell volume. For information on refining the mesh based on change in cell volume, see Gradient Adaption (p. 1688) and Volume Adaption (p. 1694).

5.2.2.4. Flow-Field Dependency

The effect of resolution, smoothness, and cell shape on the accuracy and stability of the solution process is dependent on the flow field being simulated. For example, very skewed cells can be tolerated in benign flow regions, but can be very damaging in regions with strong flow gradients.

Since the locations of strong flow gradients generally cannot be determined a priori, you should strive to achieve a high-quality mesh over the entire flow domain.

5.3. Mesh Sources

Since ANSYS Fluent can handle a number of different mesh topologies, there are many sources from which you can obtain a mesh to be used in your simulation. You can generate a mesh using ANSYS Meshing, the meshing mode of Fluent, Fluent Meshing, GAMBIT, GeoMesh, PreBFC, ICEM CFD, I-deas, NASTRAN, PATRAN, ARIES, Mechanical APDL, CFX, or other preprocessors. You can also use the mesh contained in a Fluent/UNS, RAMPANT, or FLUENT 4 case file. You can also prepare multiple mesh files and combine them to create a single mesh.

5.3.1. ANSYS Meshing Mesh Files
5.3.2. Fluent Meshing Mode Mesh Files
5.3.3. Fluent Meshing Mesh Files
5.3.4. GAMBIT Mesh Files
5.3.5. GeoMesh Mesh Files
5.3.6. PreBFC Mesh Files
5.3.7. ICEM CFD Mesh Files
5.3.8. I-deas Universal Files
5.3.9. NASTRAN Files
5.3.10. PATRAN Neutral Files
5.3.11. Mechanical APDL Files
5.3.12. CFX Files
5.3.13. Using the fe2ram Filter to Convert Files
5.3.14. Using the tpoly Filter to Remove Hanging Nodes/Edges
5.3.15. Fluent/UNS and RAMPANT Case Files
5.3.16. FLUENT 4 Case Files
5.3.17. ANSYS FIDAP Neutral Files
5.3.18. Reading Multiple Mesh/Case/Data Files
5.3.19. Reading Surface Mesh Files

5.3.1. ANSYS Meshing Mesh Files

You can use ANSYS Meshing to create your mesh. Follow the meshing procedure and export to the ANSYS Fluent mesh format (see Fluent Mesh Export in the ANSYS Meshing User's Guide for details). To

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

import the mesh into Fluent, use the **File/Read/Mesh...** ribbon tab item (as described in Reading Mesh Files (p. 44)).

5.3.2. Fluent Meshing Mode Mesh Files

You can use the meshing mode of Fluent (which was previously a stand-alone program named Fluent Meshing) to create 3D unstructured triangular/tetrahedral meshes from boundary or surface meshes. Switch to meshing mode and then follow the meshing procedure described in the Fluent Meshing User's Guide. You can save your mesh using the **File/Write/Mesh...** menu item.

Then switch to the solution mode to set up the case file for the transferred mesh, or import the saved mesh into a new Fluent session using the **File/Read/Mesh...** ribbon tab item (as described in Reading Mesh Files (p. 44)).

For information about switching between meshing and solution modes, see Switching Between Meshing and Solution Modes in the Getting Started Guide.

5.3.3. Fluent Meshing Mesh Files

Prior to version 14.5, Fluent Meshing was a stand-alone product that was used to create 3D unstructured triangular/tetrahedral meshes from boundary or surface meshes. Fluent Meshing has been integrated as the Fluent meshing mode. To import meshes created by Fluent Meshing into Fluent, use the **File/Read/Mesh...** ribbon tab item (as described in Reading Mesh Files (p. 44)).

5.3.4. GAMBIT Mesh Files

You can use GAMBIT to create 2D and 3D structured/unstructured/hybrid meshes. To create any of these meshes for ANSYS Fluent, follow the procedure described in the GAMBIT Modeling Guide, and export your mesh in Fluent 5/6 format. All such meshes can be imported directly into ANSYS Fluent using the **File/Read/Mesh...** ribbon tab item, as described in Reading Mesh Files (p. 44).

5.3.5. GeoMesh Mesh Files

You can use GeoMesh to create complete 2D quadrilateral or triangular meshes, 3D hexahedral meshes, and triangular surface meshes for 3D tetrahedral meshes. To create any of these meshes for ANSYS Fluent, follow the procedure described in the GeoMesh User's Guide.

To complete the generation of a 3D tetrahedral mesh, read the surface mesh into the meshing mode of Fluent and generate the volume mesh there (see the Fluent Meshing User's Guide for details). All other meshes can be read directly into the solution mode of Fluent. Use the **File/Read/Mesh...** ribbon tab item to read the mesh files, as described in Reading Mesh Files (p. 44).

5.3.6. PreBFC Mesh Files

You can use PreBFC to create two different types of meshes for ANSYS Fluent, structured quadrilateral/hexahedral and unstructured triangular/tetrahedral.

5.3.6.1. Structured Mesh Files

To generate a structured 2D or 3D mesh, follow the procedure described in the PreBFC User's Guide (Chapters 6 and 7). The resulting mesh will contain quadrilateral (2D) or hexahedral (3D) elements. Do not specify more than 70 wall zones and 35 inlet zones.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To import the mesh, use the **File/Import/PreBFC File...** ribbon tab item, as described in PreBFC Files (p. 68).

To manually convert a file in PreBFC format to a mesh file suitable for ANSYS Fluent, enter the following command:

`utility fl42seg` *input_filename output_filename*

The output file produced can be read into ANSYS Fluent using the **File/Read/Mesh...** ribbon tab item, as described in Reading Mesh Files (p. 44).

5.3.6.2. Unstructured Triangular and Tetrahedral Mesh Files

To generate an unstructured 2D mesh, follow the procedure described in the PreBFC User's Guide. Save the mesh file in the RAMPANT format using the `MESH-RAMPANT/TGRID` command. The current ANSYS Fluent format is the same as the RAMPANT format. The resulting mesh will contain triangular elements. To import the mesh, use the **File/Read/Mesh...** ribbon tab item, as described in Reading Mesh Files (p. 44).

To generate a 3D unstructured tetrahedral mesh, follow the procedure described in Chapter 8 of the PreBFC User's Guide for generating a surface mesh. Then read the surface mesh into the meshing mode of Fluent, and complete the mesh generation there. See the Fluent Meshing User's Guide for details.

5.3.7. ICEM CFD Mesh Files

You can use ICEM CFD to create structured meshes in FLUENT 4 format and unstructured meshes in RAMPANT format.

- To import a FLUENT 4 mesh, follow the instructions in FLUENT 4 Case Files (p. 144).
- To import a RAMPANT mesh, use the **File/Read/Mesh...** ribbon tab item, as described in Reading Mesh Files (p. 44).

The current ANSYS Fluent format is the same as the RAMPANT format, *not* the FLUENT 4 format. After reading a triangular or tetrahedral ICEM CFD volume mesh, improve its quality through smoothing (as described in Quality-Based Smoothing (p. 231)).

5.3.8. I-deas Universal Files

You can import an I-deas Universal file into ANSYS Fluent in three different ways.

- Generate an I-deas surface or volume mesh containing triangular, quadrilateral, tetrahedral, wedge and/or hexahedral elements. Import it into the meshing mode of Fluent using the commands described in the Fluent Meshing User's Guide. Adhere to the restrictions described in Appendix B: Mesh File Format of the Fluent Meshing User's Guide. In meshing mode, complete the mesh generation (if necessary); then you can switch to the solution mode of Fluent or read it into a new Fluent session as described in Fluent Meshing Mode Mesh Files (p. 136).
- Generate an I-deas volume mesh with linear triangular, quadrilateral, tetrahedral, wedge, or hexahedral elements. Import it directly using the **File/Import/I-deas Universal...** ribbon tab item, as described in I-deas Universal Files (p. 65).
- Generate an I-deas volume mesh with linear triangular, quadrilateral, tetrahedral, wedge, or hexahedral elements. Use the `fe2ram` filter to convert the Universal file to the format used by ANSYS Fluent. To convert

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

an input file in I-deas Universal format to an output file in ANSYS Fluent format, follow the instructions below in Using the fe2ram Filter to Convert Files (p. 142). After the output file is written, read it into ANSYS Fluent using the **File/Read/Mesh...** ribbon tab item, as described in Reading Mesh Files (p. 44).

5.3.8.1. Recognized I-deas Datasets

The following Universal file datasets are recognized by the ANSYS Fluent mesh import utility:

- **Node Coordinates** dataset number 15, 781, 2411
- **Elements** dataset number 71, 780, 2412
- **Permanent Groups** dataset number 752, 2417, 2429, 2430, 2432, 2435

For 2D volume meshes, the elements must exist in a constant z plane.

Note

The mesh area or mesh volume datasets are *not* recognized. This implies that writing multiple mesh areas/volumes to a single Universal file may confuse ANSYS Fluent.

5.3.8.2. Grouping Nodes to Create Face Zones

Nodes are grouped in I-deas using the `Group` command to create boundary face zones. In ANSYS Fluent, boundary conditions are applied to each zone. Faces that contain the nodes in a group are gathered into a single zone. It is important not to group nodes of internal faces with nodes of boundary faces.

One technique is to generate groups automatically based on curves or mesh areas—that is, every curve or mesh area will be a different zone in ANSYS Fluent. You may also create the groups manually, generating groups consisting of all nodes related to a given curve (2D) or mesh area (3D).

5.3.8.3. Grouping Elements to Create Cell Zones

Elements in I-deas are grouped using the `Group` command to create the multiple cell zones. All elements grouped together are placed in a single cell zone in ANSYS Fluent. If the elements are not grouped, ANSYS Fluent will place all the cells into a single zone.

5.3.8.4. Deleting Duplicate Nodes

I-deas may generate duplicate or coincident nodes in the process of creating elements. These nodes must be removed in I-deas before writing the universal file for import into ANSYS Fluent.

5.3.9. NASTRAN Files

There are three different ways in which you can import a NASTRAN file into ANSYS Fluent:

- You can generate a NASTRAN surface or volume mesh containing triangular, quadrilateral, tetrahedral, wedge, and/or hexahedral elements, and import it into the meshing mode of Fluent using the commands described in the Fluent Meshing User's Guide. Adhere to the restrictions described in Appendix B: Mesh File Format of the Fluent Meshing User's Guide. In meshing mode, complete the mesh generation (if necessary); then you can switch to the solution mode of Fluent or read it into a new Fluent session as described in Fluent Meshing Mode Mesh Files (p. 136).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You can generate a NASTRAN volume mesh with linear triangular, quadrilateral, tetrahedral, wedge, or hexahedral elements, and import it directly using the **File/Import/NASTRAN** ribbon tab item, as described in NASTRAN Files (p. 66).

- You can generate a NASTRAN volume mesh with linear triangular, quadrilateral, tetrahedral, wedge, or hexahedral elements. Then use the `fe2ram` filter to convert the NASTRAN file to the format used by ANSYS Fluent. To convert an input file in NASTRAN format to an output file in ANSYS Fluent format, follow the instructions below in Using the fe2ram Filter to Convert Files (p. 142). After the output file has been written, you can read it into ANSYS Fluent using the **File/Read/Mesh...** ribbon tab item, as described in Reading Mesh Files (p. 44).

After reading a triangular or tetrahedral NASTRAN volume mesh using the latter methods, improve its quality through smoothing (as described in Quality-Based Smoothing (p. 231)).

5.3.9.1. Recognized NASTRAN Bulk Data Entries

The following NASTRAN file datasets are recognized by the ANSYS Fluent mesh import utility:

- **GRID** single-precision node coordinates
- **GRID*** double-precision node coordinates
- **CBAR** line elements
- **CTETRA, CTRIA3** tetrahedral and triangular elements
- **CHEXA, CQUAD4, CPENTA** hexahedral, quadrilateral, and wedge elements

For 2D volume meshes, the elements must exist in a constant z plane.

5.3.9.2. Deleting Duplicate Nodes

NASTRAN may generate duplicate or coincident nodes in the process of creating elements. These nodes must be removed in NASTRAN before writing the file for import into ANSYS Fluent.

5.3.10. PATRAN Neutral Files

There are three different ways in which you can import a PATRAN Neutral file into ANSYS Fluent.

- You can generate a PATRAN surface or volume mesh containing triangular, quadrilateral, tetrahedral, wedge, and/or hexahedral elements, and import it into the meshing mode of Fluent using the commands described in the Fluent Meshing User's Guide. Adhere to the restrictions described in Appendix B: Mesh File Format of the Fluent Meshing User's Guide. In meshing mode, complete the mesh generation (if necessary); then you can switch to the solution mode of Fluent or read it into a new Fluent session as described in Fluent Meshing Mode Mesh Files (p. 136).

- You can generate a PATRAN volume mesh with linear triangular, quadrilateral, tetrahedral, wedge, or hexahedral elements (grouping nodes with the same component-group name) and import it directly to ANSYS Fluent by selecting the **File/Import/PATRAN** ribbon tab item, as described in PATRAN Neutral Files (p. 67).

- You can generate a PATRAN volume mesh with linear triangular, quadrilateral, tetrahedral, wedge, or hexahedral elements and then use the `fe2ram` filter to convert the Neutral file into the format used by ANSYS Fluent. To convert an input file in PATRAN Neutral format to an output file in ANSYS Fluent format,

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

follow the instructions below in Using the fe2ram Filter to Convert Files (p. 142). After the output file has been written, you can read it into ANSYS Fluent using the **File/Read/Mesh...** ribbon tab item, as described in Reading Mesh Files (p. 44).

After reading a triangular or tetrahedral PATRAN volume mesh using the latter methods, improve its quality through smoothing (as described in Quality-Based Smoothing (p. 231)).

Important

To retain a zone type during PATRAN Neutral file import, add the abbreviated form of the "zone-type" before the "zone-name", as shown below:

Zone Type in ANSYS Fluent	Zone Name for PATRAN export (zone-type) - (zone-name)
mass-flow-inlet	m-f-i-zone-name
pressure-outlet	p-o-zone-name
fan	fan-zone-name
velocity-inlet	v-i-zone-name
pressure-far-field	p-f-f-zone-name
solid	solid-zone-name
fluid	fluid-zone-name

5.3.10.1. Recognized PATRAN Datasets

The following PATRAN Neutral file packet types are recognized by the ANSYS Fluent mesh import utility:

- **Node Data** Packet Type 01
- **Element Data** Packet Type 02
- **Distributed Load Data** Packet Type 06
- **Node Temperature Data** Packet Type 10
- **Name Components** Packet Type 21
- **File Header** Packet Type 25

For 2D volume meshes, the elements must exist in a constant z plane.

5.3.10.2. Grouping Elements to Create Cell Zones

Elements are grouped in PATRAN using the `Named Component` command to create the multiple cell zones. All elements grouped together are placed in a single cell zone in ANSYS Fluent. If the elements are not grouped, ANSYS Fluent will place all the cells into a single zone.

5.3.11. Mechanical APDL Files

There are three different ways in which you can import a Mechanical APDL file into ANSYS Fluent.

- You can generate a surface or volume mesh containing triangular, quadrilateral, tetrahedral, wedge, or hexahedral elements using Mechanical APDL or ARIES, and import it into the meshing mode of Fluent using

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the commands described in the Fluent Meshing User's Guide. Adhere to the restrictions described in Appendix B: Mesh File Format of the Fluent Meshing User's Guide. In meshing mode, complete the mesh generation (if necessary); then you can switch to the solution mode of Fluent or read it into a new Fluent session as described in Fluent Meshing Mode Mesh Files (p. 136).

- You can generate a Mechanical APDL volume mesh with linear triangular, quadrilateral, tetrahedral, wedge, or hexahedral elements, as well as with higher order elements like 20 node hexahedron, SOLID92, and SOLID187. Then import it directly to ANSYS Fluent using the **File/Import/Mechanical APDL** ribbon tab item, as described in Mechanical APDL Files (p. 66).

  The higher order elements will be converted to their corresponding linear elements during the import in ANSYS Fluent.

- You can generate a Mechanical APDL volume mesh with linear triangular, quadrilateral, tetrahedral, wedge, or hexahedral elements, and then use the `fe2ram` filter to convert the Mechanical APDL file into the format used by ANSYS Fluent. To convert an input file in ANSYS 5.4 or 5.5 format to an output file in ANSYS Fluent format, follow the instructions in Using the fe2ram Filter to Convert Files (p. 142). After the output file has been written, you can read it into ANSYS Fluent using the **File/Read/Mesh...** ribbon tab item, as described in Reading Mesh Files (p. 44).

After reading a triangular or tetrahedral volume mesh using method 2 or 3 above, you should improve its quality through smoothing (as described in Quality-Based Smoothing (p. 231)).

5.3.11.1. Recognized ANSYS 5.4 and 5.5 Datasets

ANSYS Fluent can import mesh files from ANSYS 5.4 and 5.5 (`.cdb` files), retaining original boundary names. The following ANSYS file datasets are recognized by the ANSYS Fluent mesh import utility:

- **NBLOCK** node block data
- **EBLOCK** element block data
- **CMBLOCK** element/node grouping

The elements must be STIF63 linear shell elements. In addition, if element data without an explicit element ID is used, the filter assumes sequential numbering of the elements when creating the zones.

5.3.12. CFX Files

You can import the meshes from 3D CFX files, such as definition (`.def`) and result (`.res`) files into ANSYS Fluent, using the **File/Import/CFX** ribbon tab item, as described in CFX Files (p. 62). The `fe2ram` utility is used as the import filter, which can be used as a stand-alone program to obtain an ANSYS Fluent mesh file. See Using the fe2ram Filter to Convert Files (p. 142) for information about `fe2ram`.

Important

Note that you have the ability to import only the mesh from a CFX file, and not any results or data.

When importing a mesh from a CFX definition or results file, select whether you want the ANSYS Fluent zones to be created from the physics data objects or the primitive mesh region objects. The former option is the default and is called "Zoning by CCL" and the latter option is called "Zoning by Group".

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This will allow you to choose the type of mesh topology you would like to preserve when importing the file:

- If you want zones to be created from the physics data objects, enable **Create Zones from CCL Physics Data** in the **Select File** dialog box.
- If you want zones to be created from the primitive mesh region objects, disable **Create Zones from CCL Physics Data** in the **Select File** dialog box. This will result in the group zoning.

Important

The primitive mesh topology may contain additional regions that do not appear in the physics definition.

The default import method is the **Create Zones from CCL Physics Data** method. This method will not import CFX subdomain regions, however, the zoning by group method can import subdomain regions.

The 3D element set corresponding to zones/domains present in these files are imported as cell zones in ANSYS Fluent. They may contain tetrahedral, pyramidal, wedge, and hexahedral elements. The boundary zones in these files are a group of faces with a boundary condition name/type and are imported as face zones with the boundary condition name/type retained in ANSYS Fluent. The following boundary condition types are retained:

- **inlet**
- **outlet**
- **symmetry**
- **interface**
- **wall**

The boundaries of type `Interface` may be conformal or non-conformal. If they are non-conformal, they are retained. However, conformal interfaces contain coincident nodes that are merged and changed to type `Interior`. For some cases, for the merge to work correctly, the merge tolerance may need to be adjusted. Alternatively, the Fuse Face Zones Dialog Box (p. 2708) in ANSYS Fluent can be used to merge the conformal interfaces. For details, see Fusing Face Zones (p. 212).

5.3.13. Using the fe2ram Filter to Convert Files

The `fe2ram` filter can be used to manually convert files of certain formats into ANSYS Fluent mesh files, which can then be read into ANSYS Fluent. To use the `fe2ram` filter, enter the following at a command prompt in a terminal or command window:

`utility fe2ram` [*dimension*] *format* [*zoning*] *input_file output_file*

Note

The items enclosed in square brackets are optional. Do not type the square brackets.

- *dimension* indicates the dimension of the dataset. Replace *dimension* with `-d2` to indicate that the mesh is two dimensional. For a 3D mesh, do not enter any value for *dimension*, because 3D is the default.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- *format* indicates the format of the file you want to convert. For example, replace *format* with `-tANSYS` for a Mechanical APDL file, `-tIDEAS` for an I-deas file, `-tNASTRAN` for a NASTRAN file, and so on. To print a list of the formats which `fe2ram` can convert, type

  `utility fe2ram -cl -help`.

  Note

  fe2ram does not support VRML 2.0 or later.

- *zoning* indicates how zones were identified in the original format. Replace *zoning* by `-zID` for a mesh that was zoned by property IDs, or `-zNONE` to ignore all zone groupings. For a mesh zoned by group, do not enter anything for *zoning*, because zoning by groups is the default.

- *input_file* is the name of the original file. *output_file* is the name of the file to which you want to write the converted mesh information.

  For example, if you wanted to convert the 2D I-deas volume mesh file `sample.unv` to an output file called `sample.grd`, you will enter the following command:

  ```
  utility fe2ram -d2 -tIDEAS sample.unv sample.grd
  ```

5.3.14. Using the `tpoly` Filter to Remove Hanging Nodes/Edges

As noted in Mesh Topologies (p. 115), ANSYS Fluent can accept meshes that contain hanging nodes or hanging edges. However, the creation of interior walls can yield an error if hanging nodes/edges are located on the zone that is turned into an interior wall. This problem can occur in the following cases:

- If you read a mesh file that has an interior (or two-sided) wall boundary condition setup, but the shadow wall has not been created yet.

- If you turn an interior surface into a wall.

- If you slit an interior surface (for example, turn it into a non-conformal interface).

Such error-producing hanging nodes / edges may be present in hexcore or CutCell meshes, for example. You can remove the hanging nodes/edges by converting the associated cells to polyhedra. Each cell that is converted will retain the same overall dimensions, but the number of faces associated with the cell will increase. You can perform the conversion prior to reading the mesh into ANSYS Fluent by using the `tpoly` filter, or during an ANSYS Fluent session by using the `mesh/polyhedra/convert-hanging-nodes` text command (see Converting Cells with Hanging Nodes / Edges to Polyhedra (p. 205)).

When you use the `tpoly` filter, you must specify an input case file that contains a mesh with hanging nodes/edges. This file can either be in ASCII or Binary format, and the file should be unzipped. If the input file does not contain hanging nodes/edges, then none of the cells are converted to polyhedra.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you use the `tpoly` filter, you should specify an output case filename. After the input file has been processed by the `tpoly` filter, an ASCII output file is generated.

Important

The output case file resulting from a `tpoly` conversion only contains mesh information. None of the solver-related data of the input file is retained.

To convert a file using the `tpoly` filter, before starting ANSYS Fluent, type the following:

`utility tpoly` *input_filename output_filename*

5.3.14.1. Limitations

Meshes with polyhedra have the following limitations:

- The following mesh manipulation tools are not available on polyhedral meshes:
  - `extrude-face-zone` under the `modify-zone` option
  - skewness smoothing
  - swapping (will not affect polyhedral cells)
- The polyhedral cells that result from the conversion are not eligible for adaption. For more information about adaption, see Adapting the Mesh (p. 1681).
- The polyhedral cells that result from this conversion have the following limitations with regard to the update methods available for dynamic mesh problems:
  - When applying dynamic layering to a cell zone, you cannot have polyhedral cells adjacent to the moving face zone.
  - None of the remeshing methods except for CutCell zone remeshing will modify polyhedral cells.

  Note that smoothing is allowed for polyhedral cells, and diffusion-based smoothing is recommended over spring-based smoothing (see Diffusion-Based Smoothing (p. 650) for details). The linearly elastic solid smoothing method is not compatible with polyhedral cells.

5.3.15. Fluent/UNS and RAMPANT Case Files

If you have a Fluent/UNS 3 or 4 case file or a RAMPANT 2, 3, or 4 case file and you want to run an ANSYS Fluent simulation using the same mesh, you can read it into ANSYS Fluent using the **File/Read/Case...** ribbon tab item, as described in Reading Fluent/UNS and RAMPANT Case and Data Files (p. 53).

5.3.16. FLUENT 4 Case Files

If you have a FLUENT 4 case file and you want to run an ANSYS Fluent simulation using the same mesh, import it into ANSYS Fluent using the **File/Import/FLUENT 4 Case File...** ribbon tab item, as described

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in Fluent 4 Case Files (p. 68). ANSYS Fluent will read mesh information and zone types from the FLUENT 4 case file.

Important

FLUENT 4 may interpret some pressure boundaries differently from the current release of ANSYS Fluent. Check the conversion information printed out by ANSYS Fluent to see if you need to modify any boundary types.

To manually convert an input file in FLUENT 4 format to an output file in the current ANSYS Fluent format, enter the following command:

`utility fl42seg` *input_filename output_filename*

After the output file has been written, you can read it into ANSYS Fluent using the **File/Read/Case...** ribbon tab item, as described in Reading Mesh Files (p. 44).

5.3.17. ANSYS FIDAP Neutral Files

If you have an ANSYS FIDAP Neutral file and you want to run an ANSYS Fluent simulation using the same mesh, import it using the ribbon item, as described in ANSYS FIDAP Neutral Files (p. 64). ANSYS Fluent will read mesh information and zone types from the ANSYS FIDAP file.

To manually convert an input file in ANSYS FIDAP format to an output file in ANSYS Fluent

`utility fe2ram [` *dimension* `] -tFIDAP7` *input_file output_file*

The item in square brackets is optional. Do not type the square brackets. For a 2D file, replace *dimension* with `-d2`. For a 3D file, do not enter anything for *dimension*, because 3D is the default.

After the output file has been written, read it into ANSYS Fluent using the **File/Read/Case...** ribbon tab item, as described in Reading Mesh Files (p. 44).

5.3.18. Reading Multiple Mesh/Case/Data Files

There may be some cases in which you will need to read multiple mesh files (subdomains) to form your computational domain.

- To solve on a multiblock mesh, generate each block of the mesh in the mesh generator and save it to a separate mesh file.
- For very complicated geometries, it may be more efficient to save the mesh for each part as a separate mesh file.

The mesh node locations need not be identical at the boundaries where two separate meshes meet. ANSYS Fluent can handle non-conformal mesh interfaces. See Non-Conformal Meshes (p. 150) for details about non-conformal mesh boundaries.

There are three ways of reading multiple mesh files in ANSYS Fluent:

- Read multiple mesh files into the solution mode of Fluent.
- Read multiple mesh files into the meshing mode of Fluent.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Use `tmerge` to manipulate the individual files and merge them into a single file that can then be read into Fluent.

5.3.18.1. Reading Multiple Mesh Files via the Solution Mode of Fluent

The solution mode of Fluent allows you to handle more than one mesh at a time within the same solver settings. This capability of handling multiple meshes saves time, since you can directly read in the different mesh files and start setting up the solution, without having to switch over from meshing mode or employ another tool such as `tmerge`.

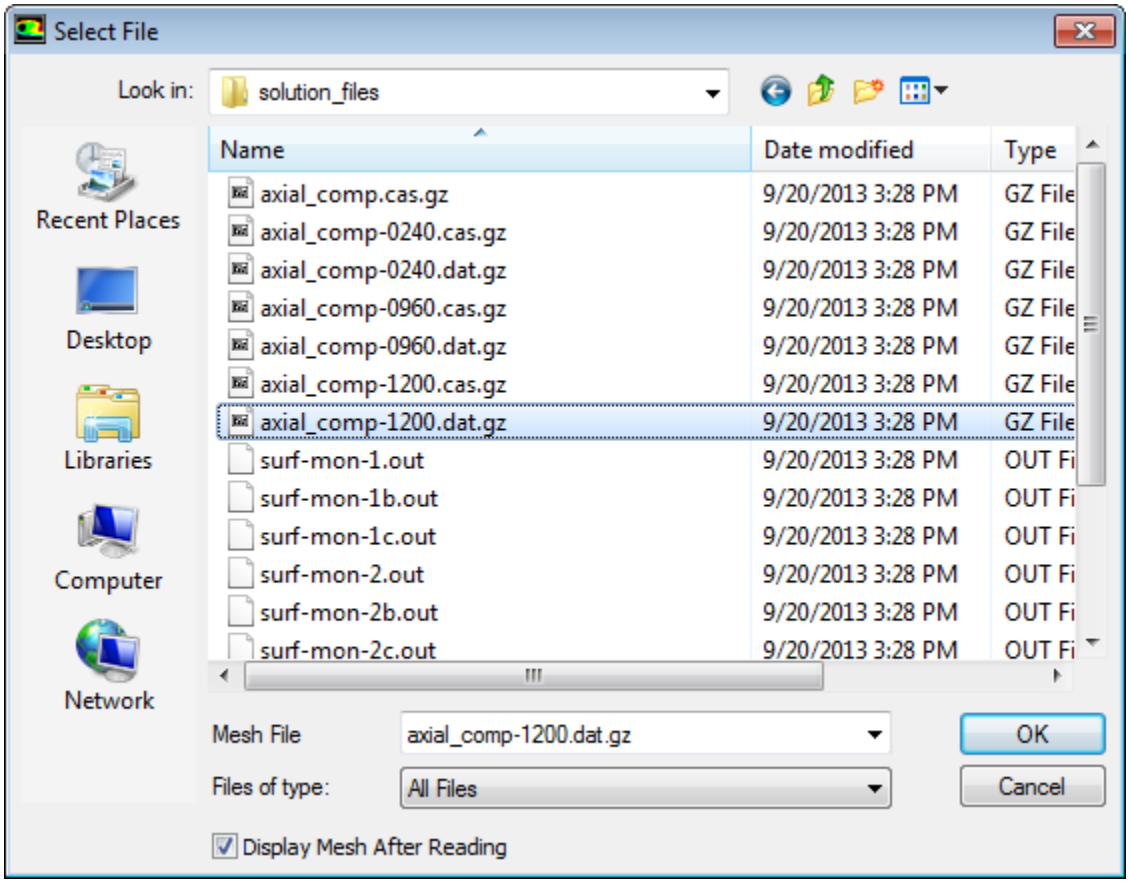
The steps to take when reading more than one mesh file are:

1. Read in your first mesh file.

   File → Read → Mesh...

   In The Select File Dialog Box (p. 19) (Figure 5.24: The Select File Dialog Box (p. 146)), select the mesh file and click **OK**.

   Figure 5.24: The Select File Dialog Box



2. Read in your second mesh file and append it to the first mesh selected in the first step.

   Setting Up Domain → Zones → Append → Append Case File...

   In The Select File Dialog Box (p. 19), select the second mesh file and click **OK**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. (optional). Display your meshes using the **Mesh Display** dialog box.

   Setup → **General** → **Display...**

   You will find that the second mesh is appended to the first.

ANSYS Fluent also allows you to append the data on the mesh. To do that, follow the procedure above. For the second step, use the following ribbon tab item:

Setting Up Domain → **Zones** → **Append** → **Append Case & Data Files...**

Select the case file in The Select File Dialog Box (p. 19) (Figure 5.24: The Select File Dialog Box (p. 146)), and click **OK**. Both the case and data files will be appended.

Important

Review your case setup after appending multiple mesh and/or data files before proceeding with the calculation.

5.3.18.2. Reading Multiple Mesh Files via the Meshing Mode of Fluent

1. Generate the mesh for the whole domain in the mesh generator, and save each cell zone (or block or part) to a separate mesh file for Fluent.

   Important

   If one (or more) of the meshes you want to import is structured (for example, a FLUENT 4 mesh file), first convert it to ANSYS Fluent format using the `fl42seg` filter described in FLUENT 4 Case Files (p. 144).

2. In the meshing mode of Fluent, combine the meshes into one mesh file.

   a. Read all of the mesh files. As the mesh files are read, they will be automatically merged into a single mesh.

   b. Save the merged mesh file.

   See the Fluent Meshing User's Guide for information about reading and writing files in meshing mode.

3. Switch over to the solution mode of Fluent or read the combined mesh file into a new session using the **File/Read/Mesh...** ribbon tab item.

For a conformal mesh, if you do not want a boundary between the adjacent cell zones, use the **Fuse Face Zones** dialog box to fuse the overlapping boundaries. For details, see Fusing Face Zones (p. 212).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The matching faces will be moved to a new zone with a boundary type of **interior** and the original zone(s) will be discarded.

Important

If you are planning to use sliding meshes, or if you have non-conformal boundaries between adjacent cell zones, do not combine the overlapping zones. Instead, change the type of the two overlapping zones to **interface** (as described in Non-Conformal Meshes (p. 150)).

5.3.18.3. Reading Multiple Mesh Files via tmerge

1. Generate the mesh for the whole domain in the mesh generator, and save each cell zone (or block or part) to a separate mesh file for ANSYS Fluent.

   Important

   If one (or more) of the meshes you want to import is structured (for example, a FLUENT 4 mesh file), first convert it to ANSYS Fluent format using the `fl42seg` filter described in FLUENT 4 Case Files (p. 144).

2. Before launching Fluent, use the `tmerge` filter to combine the meshes into one mesh file. The `tmerge` method allows you to rotate, scale, and/or translate the meshes before they are merged. Note that the `tmerge` filter allows you to merge large meshes with very low memory requirement.

   a. For 3D problems, type `utility tmerge -3d`. For 2D problems, type `utility tmerge -2d`.

   b. When prompted, specify the names of the input files (the separate mesh files) and the name of the output file in which to save the complete mesh. Be sure to include the `.msh` extension.

   c. For each input file, specify scaling factors, translation distances, and rotation information.

   For information about the various options available when using `tmerge`, type `utility tmerge -h`.

3. Read the combined mesh file into the solution mode of Fluent in the usual manner (using the **File/Read/Mesh...** ribbon tab item).

For a conformal mesh, if you do not want a boundary between the adjacent cell zones, use the **Fuse Face Zones** dialog box to fuse the overlapping boundaries. For details, see Fusing Face Zones (p. 212). The matching faces will be moved to a new zone with a boundary type of **interior** and the original zone(s) will be discarded.

Important

If you are planning to use sliding meshes, or if you have non-conformal boundaries between adjacent cell zones, do not combine the overlapping zones. Instead, change the type of the two overlapping zones to **interface** (as described in Non-Conformal Meshes (p. 150)).

In this example, scaling, translation, or rotation is not requested. Hence you can simplify the inputs to the following:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
 user@mymachine:> utility tmerge -2d

Starting /ansys_inc/v181/fluent/utility/tmerge18.1/lnamd64/tmerge_2d.18.1.0

 Append 2D grid files.
 tmerge2D ANSYS Inc, stream

 Enter name of grid file (ENTER to continue) :  my1.msh

 x,y scaling factor, eg. 1 1          : 1 1

 x,y translation, eg. 0 1             : 0 0

 rotation angle (deg), eg. 45         : 0

 Enter name of grid file (ENTER to continue) :  my2.msh

 x,y scaling factor, eg. 1 1          : 1 1

 x,y translation, eg. 0 1             : 0 0

rotation angle (deg), eg. 45          : 0

 Enter name of grid file (ENTER to continue) :  {Enter}

 Enter name of output file            : final.msh

 Reading...
  node zone: id 1, ib 1, ie 1677, typ 1
  node zone: id 2, ib 1678, ie 2169, typ 2
    .
    .
    .
 done.
 Writing...
  492 nodes, id 1, ib 1678, ie 2169, type 2.
 1677 nodes, id 2, ib 1, ie 1677, type 1.
    .
    .
    .
 done.
 Appending done.
```

5.3.19. Reading Surface Mesh Files

Surface meshes are used as background meshes for geometry-based adaption. Perform the following steps to read the surface mesh file into ANSYS Fluent:

1. Open the **Geometry Based Adaption** dialog box.

   Setting Up Domain → **Adapt** → **More** → **Geometry...**

2. Enable the **Reconstruct Geometry** option.

3. Click the **Surface Meshes...** button to open the **Surface Meshes** dialog box (Figure 5.25: The Surface Meshes Dialog Box (p. 150)).

4. In the Surface Meshes Dialog Box (p. 2807), click **Read...** and select the surface mesh file using The Select File Dialog Box (p. 19).

   Specify the units the mesh was created in (default is meters).

Note that you can also display and delete the surfaces using Figure 5.25: The Surface Meshes Dialog Box (p. 150).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.25: The Surface Meshes Dialog Box

5.4. Non-Conformal Meshes

In ANSYS Fluent it is possible to use a mesh that has non-conformal interfaces, that is, boundaries between cell zones in which the mesh node locations are not identical. Such non-conformal interfaces permit the cell zones to be easily connected to each other by passing fluxes from one mesh to another. The principle requirement is that the boundary zones that make up the non-conformal interface must overlap either partially or fully. (This requirement does not apply to non-conformal periodic boundaries.)

5.4.1. Non-Conformal Mesh Calculations
5.4.2. Non-Conformal Interface Algorithm
5.4.3. Requirements and Limitations of Non-Conformal Meshes
5.4.4. Using a Non-Conformal Mesh in ANSYS Fluent

5.4.1. Non-Conformal Mesh Calculations

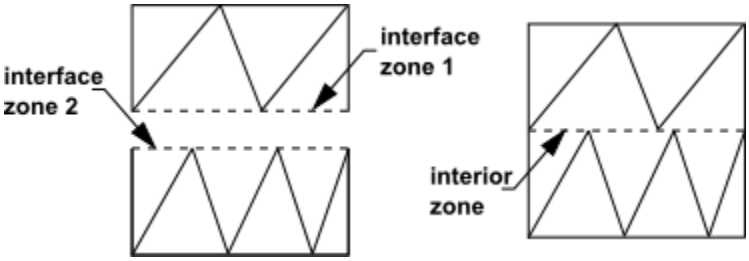
To compute the flux across the non-conformal boundary, ANSYS Fluent must first compute the intersection between the interface zones that comprise the boundary. In the case of fluid-to-fluid zone interfaces and solid-to-solid zone interfaces, the resulting intersection produces an interior zone where the two interface zones overlap (see Figure 5.26: Completely Overlapping Mesh Interface Intersection (p. 151)). In the case of fluid-to-solid zone interfaces, a coupled wall zone is produced where the interface zones overlap (see The Coupled Wall Option (p. 156)).

Note

If the zones on either side of a solid-to-solid interface are of differing materials, the interface must be treated as a coupled wall instead of as an interior. If Fluent detects this scenario during case read or initialization it will issue a warning and prompt you to run the `mesh/modify-zones/slit-interior-between-diff-solids` TUI command.

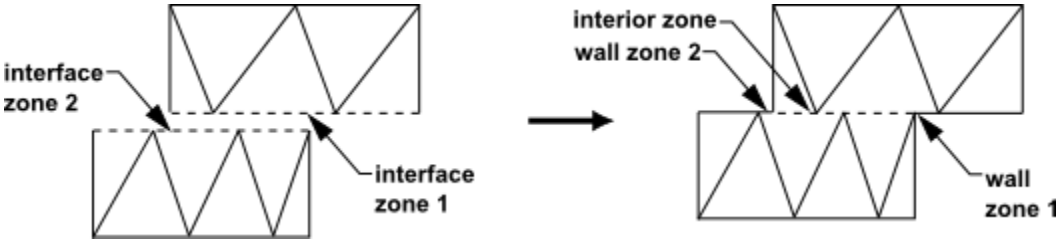
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.26: Completely Overlapping Mesh Interface Intersection



If one of the interface zones extends beyond the other (Figure 5.27: Partially Overlapping Mesh Interface Intersection (p. 151)), by default ANSYS Fluent will create additional wall zones for the portion(s) of the boundary where the two interface zones do not overlap.

Figure 5.27: Partially Overlapping Mesh Interface Intersection

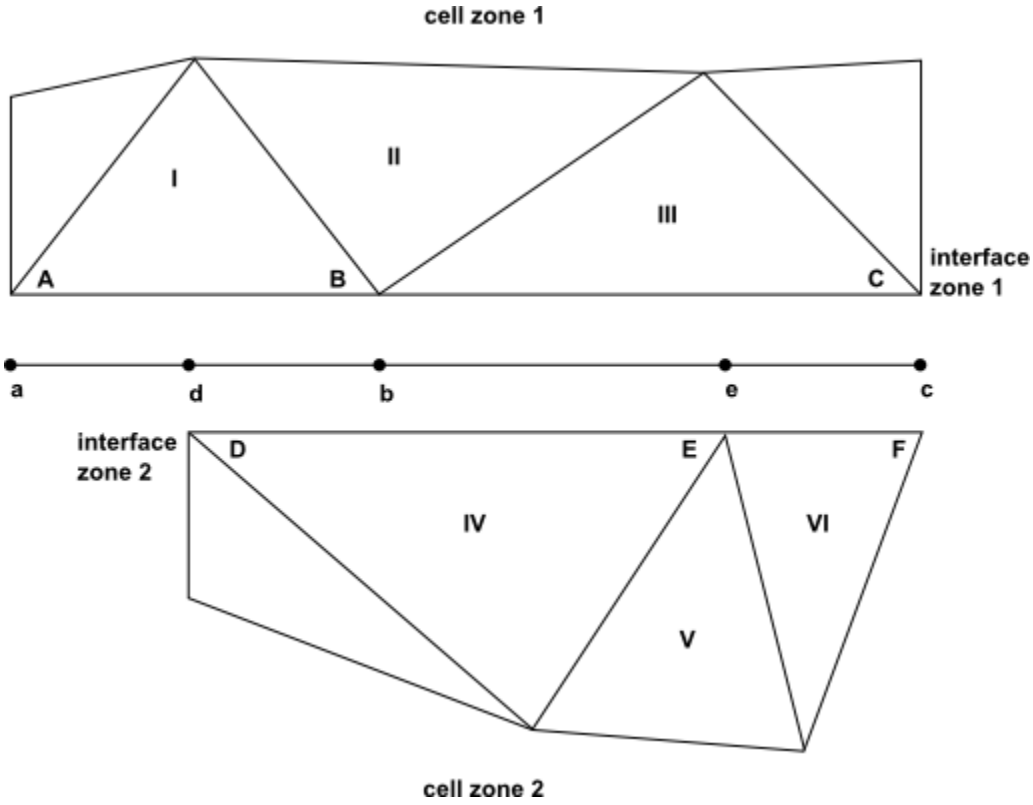


Fluxes across the mesh interface are computed using the faces resulting from the intersection of the two interface zones, not from the interface zone faces.

In the example shown in Figure 5.28: Two-Dimensional Non-Conformal Mesh Interface (p. 152), the interface zones are composed of faces A-B and B-C, and faces D-E and E-F.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.28: Two-Dimensional Non-Conformal Mesh Interface



The intersection of these zones produces the faces a-d, d-b, b-e, and e-c. Faces produced in the region where the two cell zones overlap (d-b, b-e, and e-c) are grouped to form an interior or coupled wall zone (depending on the types of the adjacent cell zones), while the remaining face (a-d) forms a wall zone.

To compute the flux across the interface into cell IV, face D-E is ignored and instead faces d-b and b-e are used to bring information into cell IV from cells I and III.

While the previous discussion described the default treatment of a non-conformal interface, there are several options you can enable at the interface in order to revise the treatment of the fluxes and/or reduce the memory usage and processing time:

- periodic boundary condition
- periodic repeats
- coupled wall
- matching
- mapped
- static

These non-conformal interface options are described in the following sections.

5.4.1.1. The Periodic Boundary Condition Option
5.4.1.2. The Periodic Repeats Option
5.4.1.3. The Coupled Wall Option
5.4.1.4. Matching Option

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.4.1.5. The Mapped Option
5.4.1.6. The Static Option
5.4.1.7. Interface Zones Automatic Naming Conventions

5.4.1.1. The Periodic Boundary Condition Option

Non-conformal interfaces can be used to implement a periodic boundary condition like that described for conformal periodic boundaries (see Periodic Boundary Conditions (p. 386)). The advantage of using a mesh interface is that, unlike the standard periodic boundary condition, the nodes of the two zones do not have to match one-for-one.

The interface zones that utilize the periodic boundary condition option (Figure 5.29: Non-Conformal Periodic Boundary Condition (Translational) (p. 153) and Figure 5.30: Non-Conformal Periodic Boundary Condition (Rotational) (p. 154)) are coupled in the manner described in the previous section, except that the zones do not overlap (that is, the zones are not spatially coincident at any point). In order to generate the new faces that will be used to compute the fluxes across the interface, the nodes of the first zone are either translated or rotated (about a given axis) onto the other zone. The distance/angle that the nodes are translated/rotated is called the "periodic offset". The new faces will be defined between all of the combined nodes, and then applied to each of the original zones.

Figure 5.29: Non-Conformal Periodic Boundary Condition (Translational)

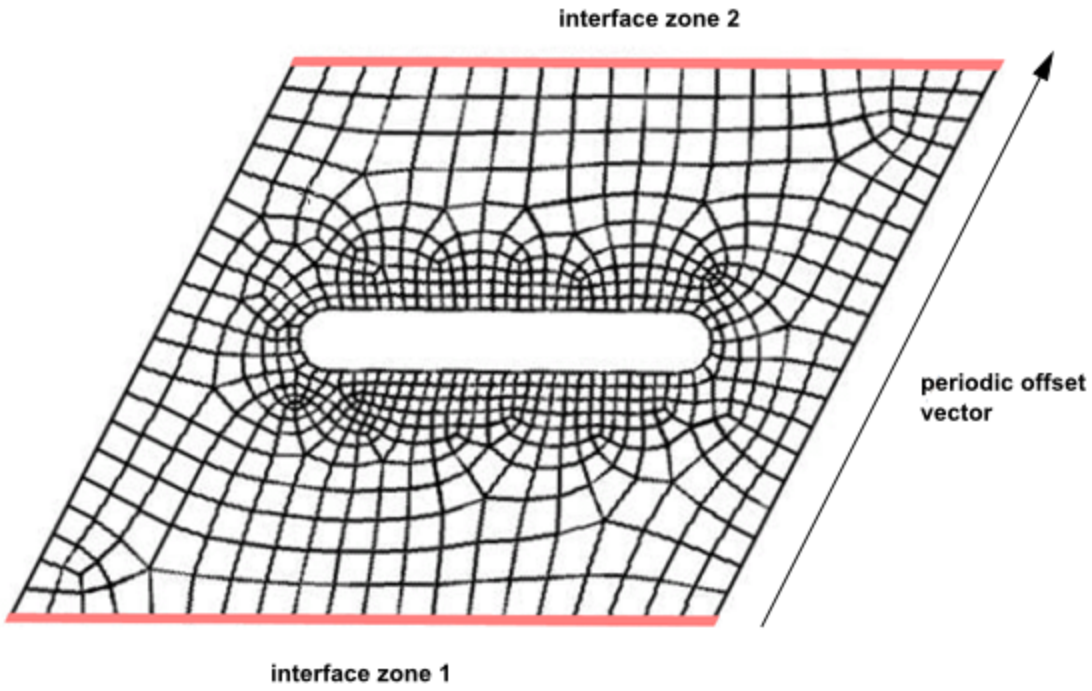
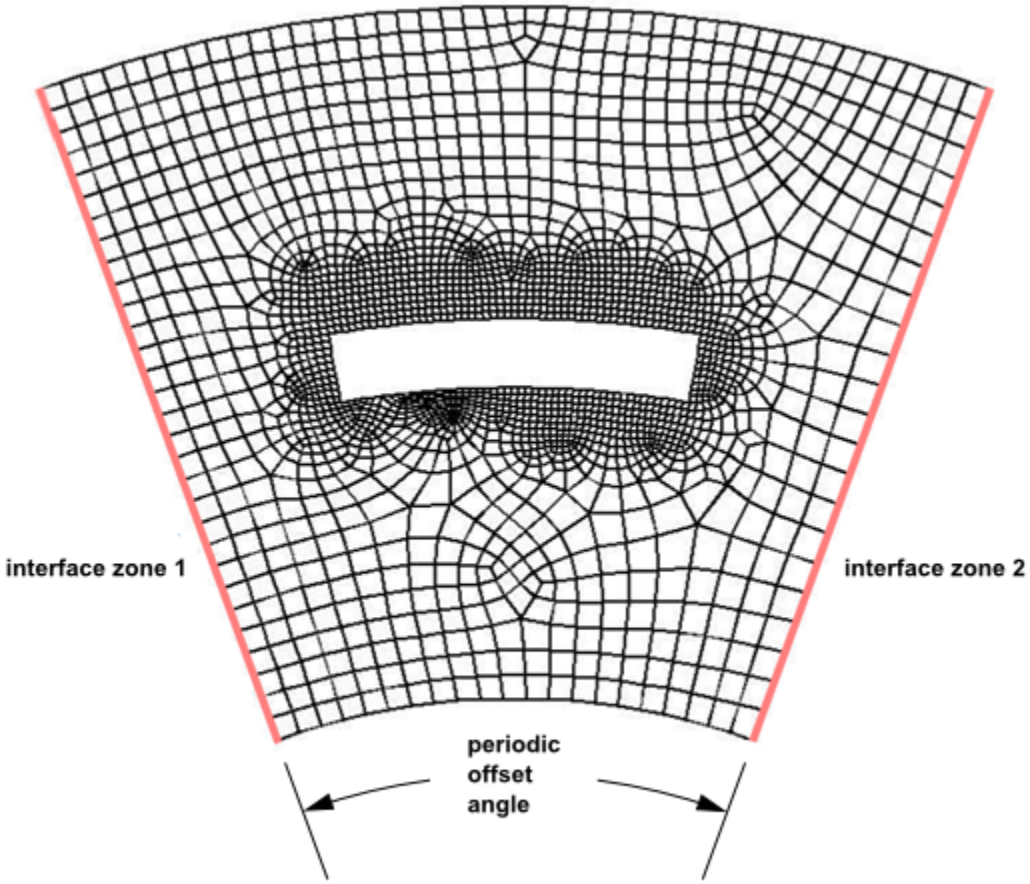


Figure 5.30: Non-Conformal Periodic Boundary Condition (Rotational)



5.4.1.2. The Periodic Repeats Option

The periodic repeats option is appropriate when each of the interface zones is adjacent to a non-conformal periodic interface or a pair of conformal periodic zones (see Figure 5.31: Translational Non-Conformal Interface with the Periodic Repeats Option (p. 155) and Figure 5.32: Rotational Non-Conformal Interface with the Periodic Repeats Option (p. 156)). The periodic repeats option takes into account the repeating nature of the flow solutions in the two cell zones in the following manner. Wherever the interface zones overlap (that is, wherever interface zone 1 and 2 are spatially coincident), the fluxes on either side of the interface are coupled in the usual way. The portion of interface zone 1 that does not overlap is coupled to the non-overlapping portion of interface zone 2, by translating or rotating the fluxes by the periodic offset. This is similar to the treatment of non-conformal periodic boundary conditions. The periodic repeats option is typically used in conjunction with the sliding mesh model when simulating the interface between a rotor and stator.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.31: Translational Non-Conformal Interface with the Periodic Repeats Option

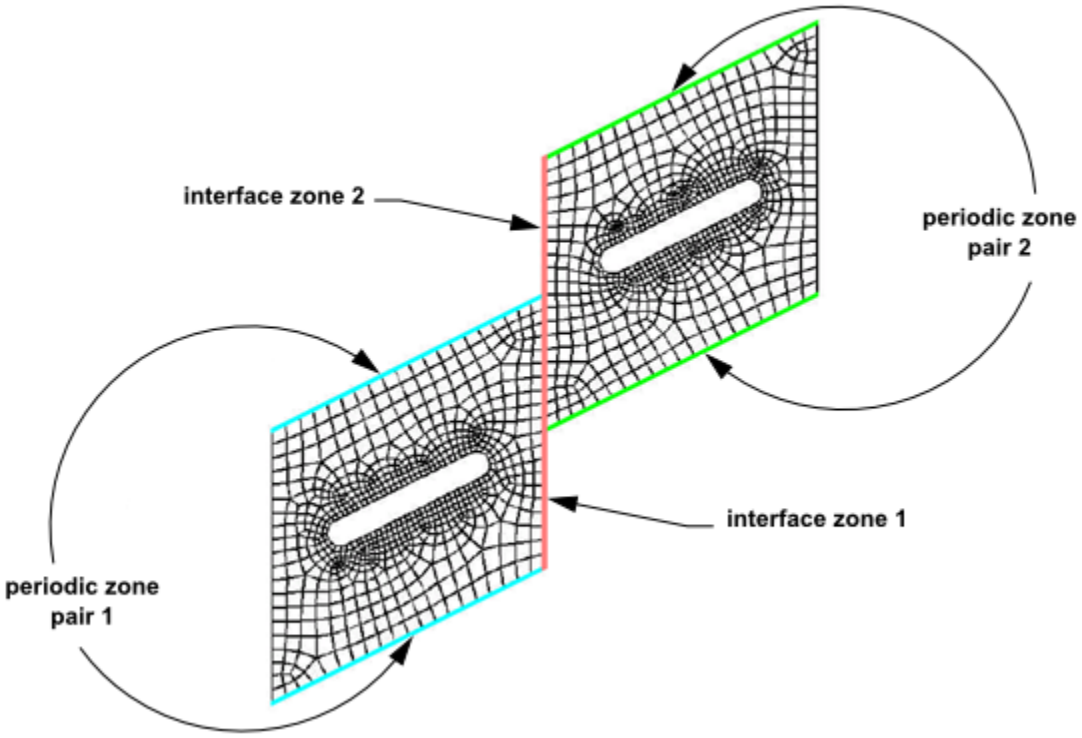
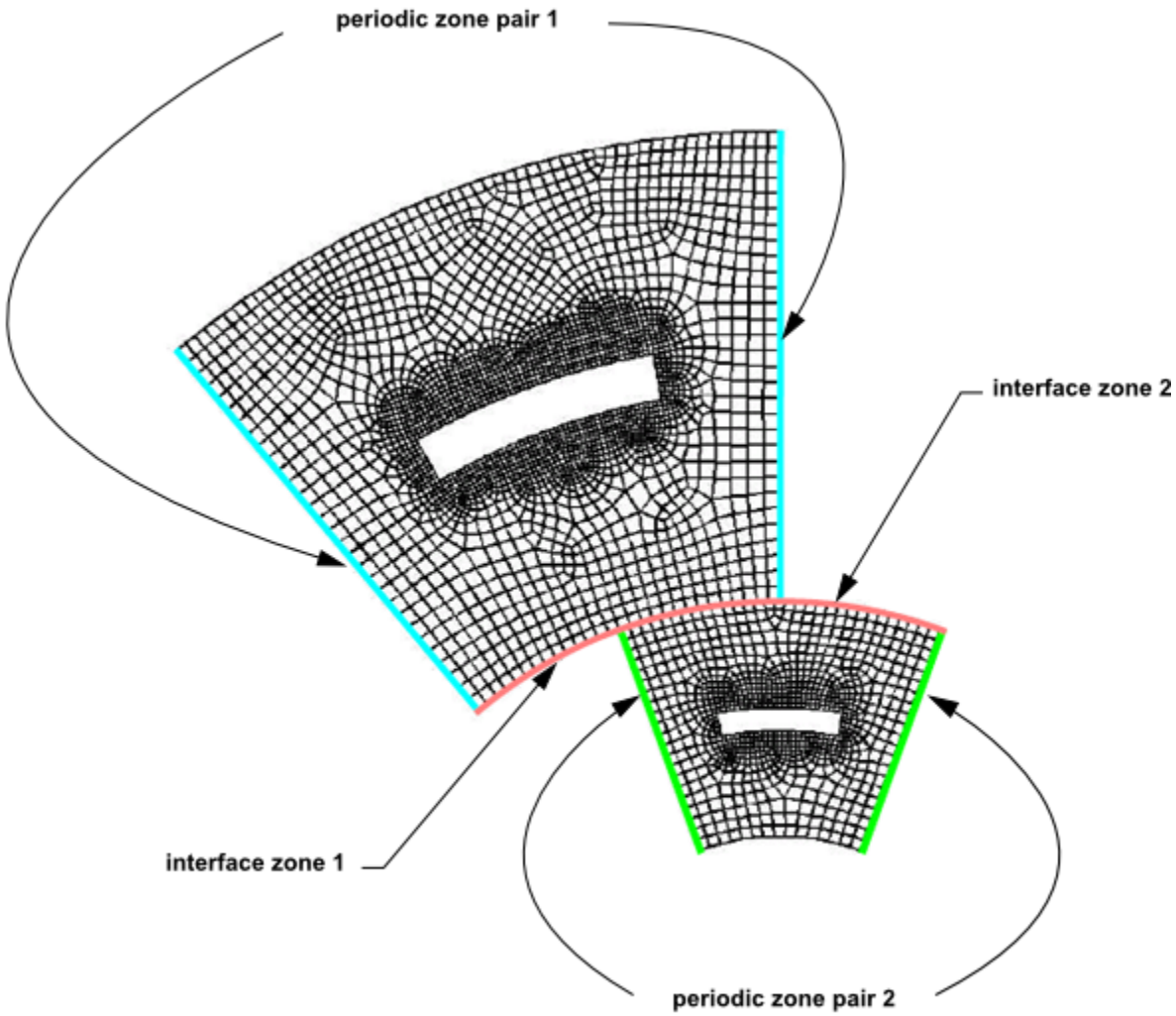


Figure 5.32: Rotational Non-Conformal Interface with the Periodic Repeats Option



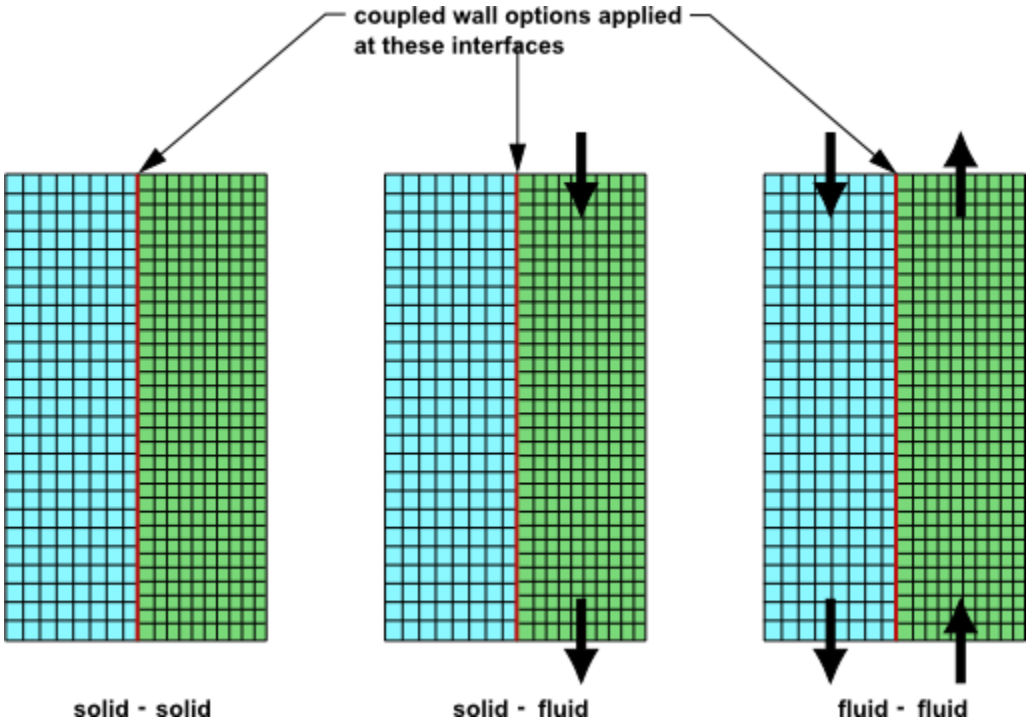
5.4.1.3. The Coupled Wall Option

As described previously, the typical function of non-conformal interfaces is to couple fluid zones, so as to permit fluid flow to pass from one mesh interface to the other. Another available option is to create a coupled wall boundary at the interface. In such a case, fluid flow would not pass across the interface, as the interface is acting as a wall zone. Coupled wall heat transfer, on the other hand, would be permitted. Such an interface is required if one or both of the cell zones is a solid. It is also allowable if both of the cell zones are fluids; for example, you can model a thin wall or baffle separating the two fluid zones. Figure 5.33: Non-Conformal Coupled Wall Interfaces (p. 157) illustrates coupled walls with both solid and fluid zones.

Note that coupled walls can also make use of the periodic repeats option. That is, both options can be invoked simultaneously. For details see Using a Non-Conformal Mesh in ANSYS Fluent (p. 164).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.33: Non-Conformal Coupled Wall Interfaces



5.4.1.4. Matching Option

When two interface zones do not match well (which could result from different mesh topologies, geometric misalignment, or the existence of large gaps), wall zones will be created on the interface. If, in such cases, you want to have only internal zones generated on the interface (that is, interior or wall / shadow pairs), it is best to use a matching interface. If the interface zones are completely overlapping (see Figure 5.26: Completely Overlapping Mesh Interface Intersection (p. 151)) as opposed to partially overlapping (see Figure 5.27: Partially Overlapping Mesh Interface Intersection (p. 151)) and so no wall zones are expected to be created on the mesh interface, it is recommended that you select the **Matching** option.

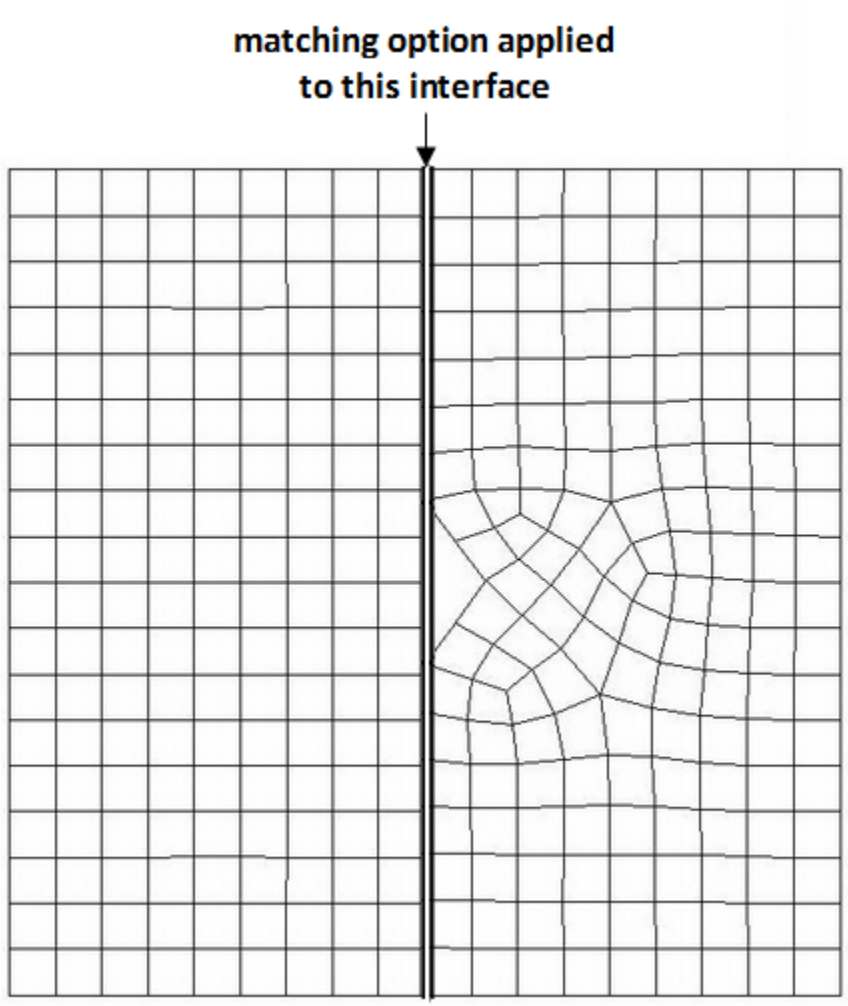
Note

The **Matching** option is also compatible with the periodic boundary condition option.

Figure 5.34: Matching Non-Conformal Wall Interfaces (p. 158) below shows two interface zones that do not match up well in the center -- the interface zone on the right has an inverted spike. Even with such a large mismatch, the **Matching** option still enables you to create an interface with only the interior zone.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.34: Matching Non-Conformal Wall Interfaces



When the **Matching** option is selected, ANSYS Fluent checks if one interface zone sits on top of the other and displays messages if there is an overlap of cell zones that are connected through a mesh interface (therefore making the interface invalid for use). Since the severity of zone overlap is different for matching interfaces and non-matching interfaces, ANSYS Fluent displays different messages accordingly.

For example, for non-matching interfaces:

```
Info: Interface zones overlap for mesh interface (interface-name). This could adversely affect your solution.
```

Likewise, for matching interfaces:

```
Warning: Interface zone overlaps for mesh interface (interface-name).
         This could generate left-handed faces on the interface and make it invalid for use.
```

5.4.1.5. The Mapped Option

The **Mapped** option is an alternative approach for modeling coupled walls between zones. This approach is more robust than the standard non-conformal interface formulations when the interface zones penetrate each other or have gaps between them. The **Mapped** option is less accurate, but it is useful in situations where using the standard non-conformal interface formulations causes the simulation to fail (as may occur when there is interface penetration or gaps between interfaces, causing the creation of left-handed faces at the interface).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.35: Non-Conformal Mapped Interface with a Gap and Penetration

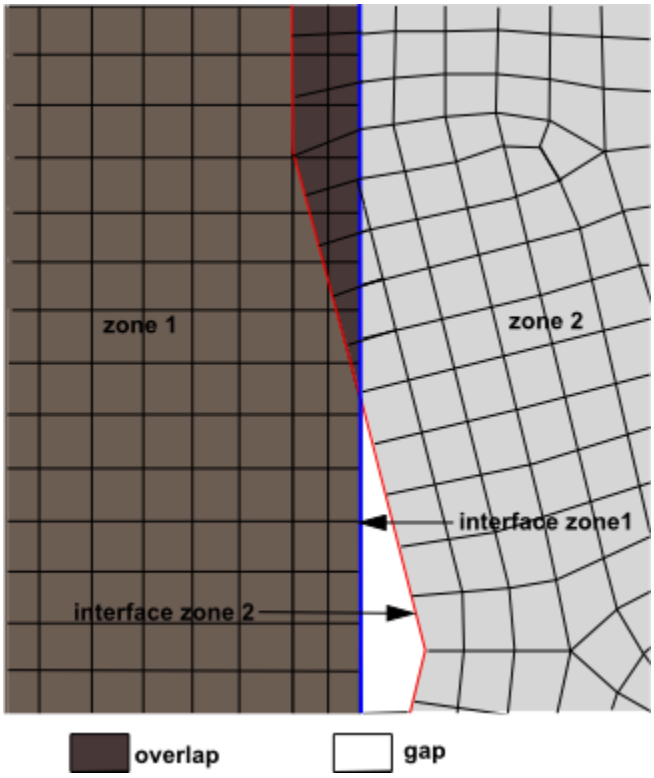


Figure 5.35: Non-Conformal Mapped Interface with a Gap and Penetration (p. 159) shows significant penetration between the two interface zones and large gap. The mapped option allows you to create a coupled wall even with the large amount of penetration and gap shown in this image.

The **Mapped** option only works in conjunction with the **Coupled Wall** option. Enabling the **Mapped** automatically enables **Coupled Wall**.

You can use the Interface Options Dialog Box (p. 2717) to modify the global settings for the mapped interfaces, as well as to convert existing mesh interfaces (either all of them or just those that have penetrating zones) to use the **Mapped** option.

You can confirm that you created good quality mapped interfaces by performing a mesh check. If you have mapped interfaces that did not pass the quality check, Fluent will suggest that you use the text command `define/mesh-interface/improve-quality`, which allows you to increase the interface tolerance to improve the quality of the mapped interfaces.

Note the following limitations when using the **Mapped** option:

- The **Mapped** option is not compatible with shell conduction.
- Thermal coupling calculations will not be performed at mapped interfaces for the Eulerian multiphase model.
- No other interface options besides **Coupled Wall** are allowed with the **Mapped** option.
- For the S2S model, the **Face to Face** basis for the view factors is not compatible with mapped interfaces for polyhedral meshes.
- Mapped interfaces are not supported with the density-based solver.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Mapped interfaces are not supported with the Lagrangian discrete phase model.
- Mapped interfaces are not supported with moving / deforming meshes.

5.4.1.6. The Static Option

For a default mesh interface or when the **Coupled** and/or **Matching** options are enabled, ANSYS Fluent will automatically create zones that will be needed if the interface should change due to motion (from moving reference frames or sliding meshes) or deformation. If you know in advance that the interface zones will not be moving or deforming relative to each other at the interface, then (and only then) it is recommended that you enable the **Static** option. The **Static** option prevents the creation of such zones, and thus reduces the memory usage and processing time (for both the creation of the interface and the running of the calculation). This reduction can be significant, especially when there are many zones on both sides of the interface.

> **Warning**
>
> If the **Static** option is applied at an interface that does end up undergoing relative motion or deformation, the results will be incorrect.

Note that the **Static** option is not limited to steady-state cases.

5.4.1.7. Interface Zones Automatic Naming Conventions

The following sub-sections describe the automatic naming conventions for each type of non-conformal interface:

5.4.1.7.1. Default (No Options Enabled)
5.4.1.7.2. Periodic Boundary Condition
5.4.1.7.3. Periodic Repeats
5.4.1.7.4. Coupled Wall
5.4.1.7.5. Matching
5.4.1.7.6. Mapped
5.4.1.7.7. Static

5.4.1.7.1. Default (No Options Enabled)

The automatic naming convention for interface boundary zones is as follows: *<mesh interface name> (user-provided) - side (1 or 2) - wall - <interface zone name>*.

The automatic naming convention for interface wall zones is as follows: *<mesh interface name> (user-provided) - wall1 (only wall1, because the other wall is the shadow of wall1) - <connection number> (1–m, where m is the number of zones in interface zone side 1) - <connection number> (1–n, where n is the number of zones in interface zone side 2) - shadow (for interface wall zone side 2 only)*.

The automatic naming convention for interface interior zones is as follows: *<mesh interface name> (user-provided) - interior - <connection number> (1–m, where m is the number of zones in interface zone side 1) - <connection number> (1–n, where n is the number of zones in interface zone side 2)*.

5.4.1.7.2. Periodic Boundary Condition

The automatic naming convention for interface boundary zones is as follows: *<mesh interface name> (user-provided) - side (1 or 2) - wall - <interface zone name>*.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The automatic naming convention for interface interior zones is as follows: *<mesh interface name> (user-provided) - periodic*.

5.4.1.7.3. Periodic Repeats

The automatic naming convention for interface interior zones is as follows: *<mesh interface name> (user-provided) - periodic*.

5.4.1.7.4. Coupled Wall

The automatic naming convention for interface boundary zones is as follows: *<mesh interface name> (user-provided) - side (1 or 2) - wall - <interface zone name>*.

The automatic naming convention for interface wall zones is as follows: *<mesh interface name> (user-provided) - wall1 (only wall1, because the other wall is the shadow of wall1) - <connection number> (1–m, where m is the number of zones in interface zone side 1) - <connection number> (1–n, where n is the number of zones in interface zone side 2) - shadow (for interface wall zone side 2 only)*.

For example,

```
> define/mesh-interfaces/list

List of Mesh Interfaces


Interface Name:     cw:01
Interface Options:  Coupled


Description                    Name                    ID          Area          Area Percentage
---------------------  ------------------------------  ----------  ------------------  -----------------
      Interface-Zone-1                             a2          17        0.1707190424
      Interface-Zone-1                             a1          13        0.1707191467
      Interface-Zone-2                             b1           4        36.7499923706
       Boundary-Zone-2             cw:01-side2-wall-b1          53        36.4085540771         99.070915%
 Interface-Wall-Zone-1                 cw:01-wall1-1-1          50        0.1706306934         99.948250%
 Interface-Wall-Zone-1                 cw:01-wall1-2-1          49        0.1706308126         99.948257%
 Interface-Wall-Zone-2          cw:01-wall1-1-1-shadow          54        0.1706306934          0.464301%
 Interface-Wall-Zone-2          cw:01-wall1-2-1-shadow          55        0.1706308126          0.464302%
```

5.4.1.7.5. Matching

The automatic naming convention for interface interior zones is as follows: *<mesh interface name> (user-provided) - interior - <connection number> (1–m, where m is the number of zones in interface zone side 1) - <connection number> (1–n, where n is the number of zones in interface zone side 2)*.

5.4.1.7.6. Mapped

The automatic naming convention for interface boundary zones is as follows: *<mesh interface name> (user-provided) - side (1 or 2) - wall - <interface zone name>*.

The automatic naming convention for interface wall zones is as follows: *<mesh interface name> (user-provided) - wall (1 or 2) - <connection number> (1–m, where m is the number of zones in interface zone side 1) - <connection number> (1–n, where n is the number of zones in interface zone side 2)*.

For example,

```
> define/mesh-interfaces/list

List of Mesh Interfaces
```

```
Interface Name:     map:01
Interface Options:  Mapped

Description                        Name                    ID            Area          Area Percentage
---------------------  ------------------------------  ----------  ------------------  -----------------
      Interface-Zone-1                            a2          17        0.1707190424
      Interface-Zone-1                            a1          13        0.1707191467
      Interface-Zone-2                            b1           4        36.7499923706
       Boundary-Zone-2           map:01-side2-wall-b1          32        36.4085540771         99.070915%
 Interface-Wall-Zone-1                map:01-wall1-a2          28        0.1707190424         100.000000%
 Interface-Wall-Zone-1                map:01-wall1-a1          27        0.1707191467         100.000000%
 Interface-Wall-Zone-2                map:01-wall2-b1          29        0.3414205313           0.929036%
```

5.4.1.7.7. Static

The **Static** option does not affect the naming conventions of the zones; that is, the zone names will be based entirely on the other settings in the **Interface Options** group box.

5.4.2. Non-Conformal Interface Algorithm

In the current version of ANSYS Fluent, non-conformal interface calculations are handled using one of three approaches depending on the type of interface. **Coupled Wall** interfaces are handled using interface boundary metrics and **Mapped** interfaces use a projection approach. The interface boundary metrics and projection approach are both robust algorithms that avoid creating left-handed faces and are particularly useful when you have complex geometries.

The remaining types of non-conformal interfaces are handled using a virtual polygon approach. The virtual polygon approach stores the area vector and centroid of the polygon faces. This approach does not involve node movement and cells are not necessarily water-tight cells. Hence gradients are corrected to take into account the missing face area. Note that in transient cases, the stationary non-conformal interfaces will be automatically preserved by ANSYS Fluent during mesh update.

Previous versions (16.0 and earlier) of Fluent used the virtual polygon approach for all non-conformal interfaces. The interface boundary metrics is a more robust approach for creating coupled wall interfaces than the virtual polygon approach because it avoids creating left-handed faces. Older versions of Fluent (6.1 or earlier) used a triangular face approach, which triangulated the polygon intersection faces and stored triangular faces. This approach involved node movement and water-tight cells, and was not as stable as the virtual polygon approach. Note that case files created in releases prior to 16.0 can be read and run normally in the current version of Fluent unless you recreate any of the interfaces. Recreated coupled wall interfaces will use interface boundary metrics and all other recreated interfaces will use the virtual polygon approach.

It is possible that distorted meshes may be produced during sliding mesh calculations, generating left-handed faces. You cannot obtain a flow solution until all of the faces are "right handed", and so ANSYS Fluent corrects the left-handedness of these faces automatically. In extreme cases, the left-handed faces cannot be fully corrected and are deleted automatically, so that the solution does not diverge.

Left-handed cells can also be created for the geometries that contain sharp edges and corners, which may affect the final solution. For such geometries, you should first separate the zones and then create the interfaces separately to get the better solution.

The additional input of the angle/translation vector at the `angle/translation-vector` prompt in the console may be required to recreate face-periodic interfaces. Also, with the current mesh interface algorithm in parallel, there is no need for encapsulation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.4.3. Requirements and Limitations of Non-Conformal Meshes

This section describes the requirements and limitations of non-conformal meshes:

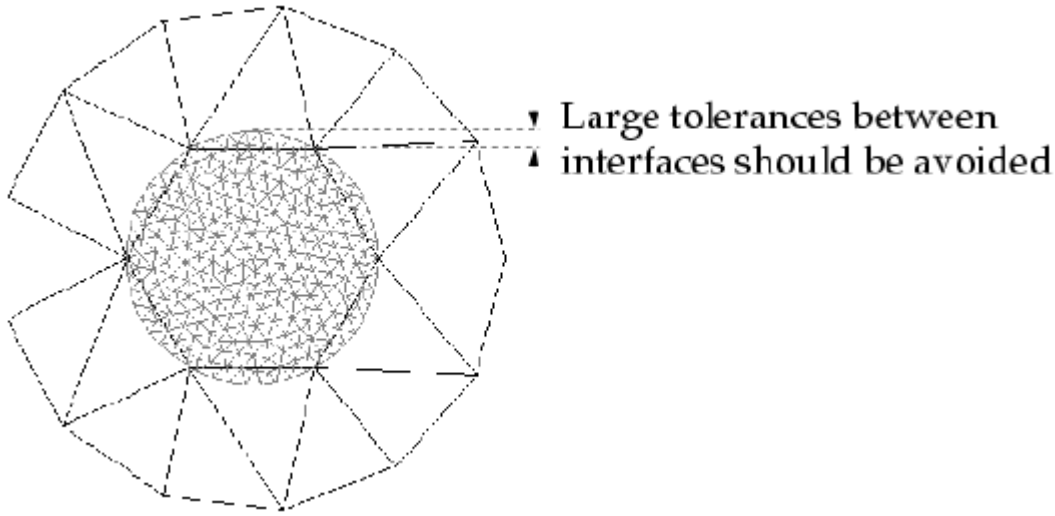
- The mesh interface can be of any shape (including a non-planar surface, in 3D), provided that the two interface boundaries are based on the same geometry. If there are sharp features (for example, 90-degree angles) or curvature in the mesh, it is especially important that both sides of the interface closely follow that feature.

  For example, consider the case of two concentric circles that define two fluid zones with a circular, non-conformal interface between them, as shown in Figure 5.36: A Circular Non-Conformal Interface (p. 163). Because the node spacing on the interface edge of the outer fluid zone is coarse compared to the radius of curvature, the interface does not closely follow the feature (in this case, the circular edge.)

  Important

  The maximum tolerance between two interfaces should not be larger than their adjacent cell size at that location That is no cell should be completely enclosed between two interfaces.

Figure 5.36: A Circular Non-Conformal Interface



- If you create a single mesh with multiple cell zones separated by a non-conformal boundary, you must be sure that each cell zone has a distinct face zone on the non-conformal boundary.

  The face zones for two adjacent cell zones will have the same position and shape, but one will correspond to one cell zone and one to the other. It is also possible to create a separate mesh file for each of the cell zones, and then merge them as described in Reading Multiple Mesh/Case/Data Files (p. 145).

- All periodic zones must be correctly oriented (either rotational or translational) before you create the non-conformal interface.

- In order for the periodic boundary condition option or periodic repeats option to be valid, the edges of the second interface zone must be offset from the corresponding edges of the first interface zone by a uniform

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

amount (either a uniform translational displacement or a uniform rotation angle). This is not true for non-conformal interfaces in general.

The periodic boundary condition option has the additional requirement that the angle associated with a rotational periodic must be able to divide 360 without remainder.

- The periodic repeats option requires that some portion of the two interface zones must overlap (that is, be spatially coincident).
- The periodic repeats option requires that the non-overlapping portions of the interface zones must have identical shape and dimensions. If the interface is part of a sliding mesh, you must define the mesh motion such that this criterion is met at all times.
- Note that for 3D cases, you cannot have more than one pair of conformal periodic zones adjacent to each of the interface zones.
- You must not have a single non-conformal interface where part of the interface is made up of a coupled two-sided wall, while another part is not coupled (that is, the normal interface treatment). In such cases, you must break the interface up into two interfaces: one that is a coupled interface, and the other that is a standard fluid-fluid interface. See Using a Non-Conformal Mesh in ANSYS Fluent (p. 164) for information about creating coupled interfaces.
- For simulations that involve the Fluent, Mechanical, and Meshing applications, meshing problems can arise in instances where there are multiple regions and contacts between them. In Fluent, a zone can only exist in a single contact region. The Mechanical and Meshing applications both use a different approach concerning contact regions when compared to Fluent.
- Fluid zones designated as 3D fan zones cannot have non-conformal interfaces.

5.4.4. Using a Non-Conformal Mesh in ANSYS Fluent

If your multiple-zone mesh includes non-conformal boundaries, check if the mesh meets all the requirements (listed in Requirements and Limitations of Non-Conformal Meshes (p. 163)). This ensures that ANSYS Fluent can obtain a solution on the mesh. Then do the following:

1. Read the mesh into ANSYS Fluent. If you have multiple mesh files that have not yet been merged, first follow the instructions in Reading Multiple Mesh/Case/Data Files (p. 145) to merge them into a single mesh.
2. For all of the zones that comprise the non-conformal boundaries, change the type to **interface** (as described in Changing Cell and Boundary Zone Types (p. 239)).

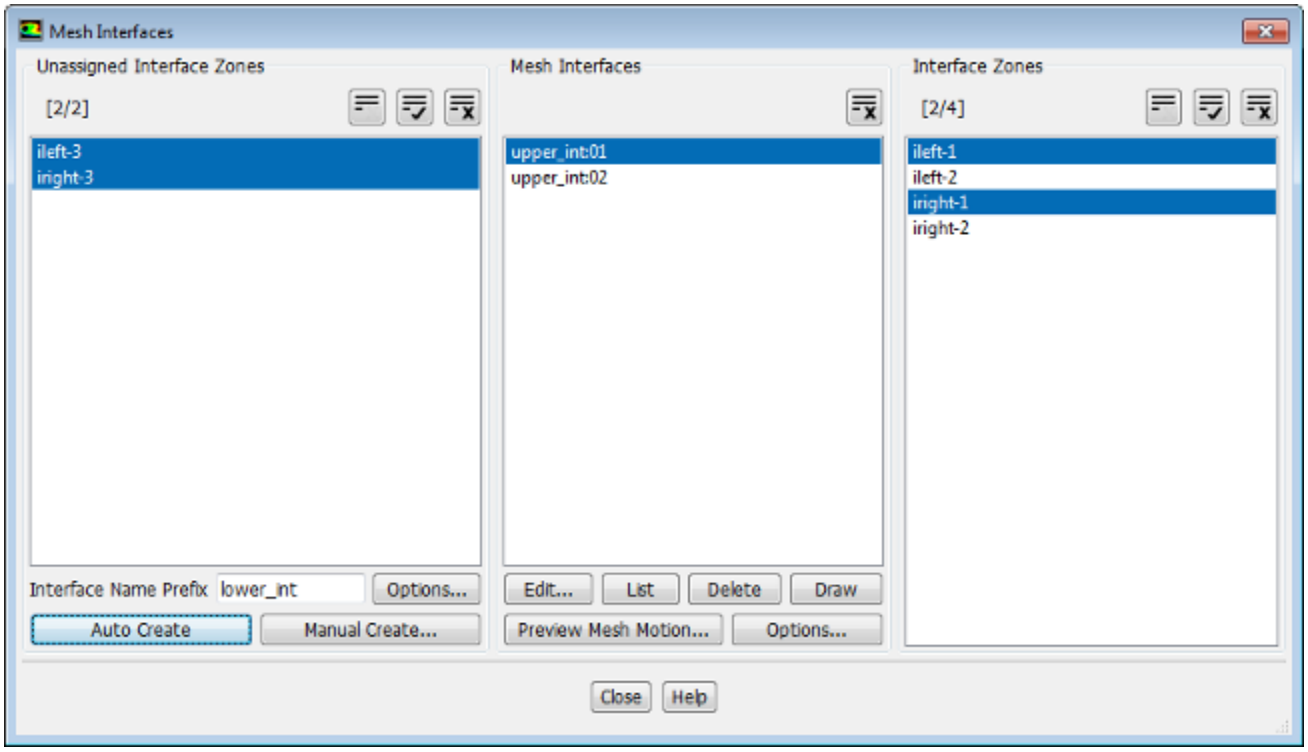
   Setup → **Boundary Conditions**

3. Open the Mesh Interfaces Dialog Box (p. 2732) (Figure 5.37: The Mesh Interfaces Dialog Box (p. 165)).

   Setup → **Mesh Interfaces** → **New...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.37: The Mesh Interfaces Dialog Box



4. Create all of the mesh interfaces that use the periodic or periodic repeat option. This must be done using the manual method, by clicking the **Manual Create...** button and using the Create/Edit Mesh Interfaces Dialog Box (p. 2680). For details, see Manually Creating Mesh Interfaces (p. 170).

5. Create all of the mesh interfaces that *do not* use the periodic or periodic repeat option, using the automatic method. As part of this method, Fluent will automatically determine which of the selected interface boundary zones can be grouped together to form the two sides of one or more mesh interfaces.

   To use the automatic method, perform the following steps in the **Mesh Interfaces** dialog box:

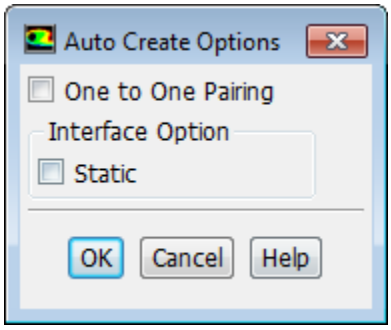
   a. Select all of the zones in the selection list in the **Unassigned Interface Zones** group box. This list contains only those zones that are not already part of an existing mesh interface. Note that you may want to select only some of the zones when you want the resulting mesh interfaces to have different name prefixes or different settings in the **Auto Create Options** dialog box (as described in the steps that follow); for example, you could select only the interface zones from the front half of a model, and then have all of the resulting mesh interfaces named with the prefix "front".

   b. Enter the **Interface Name Prefix** you want applied to all of the resulting interfaces.

   c. (optional) Click the **Options...** button in the **Unassigned Interface Zones** group box to open the Auto Create Options Dialog Box (p. 2672), and enable the settings that you want applied to all of the resulting interfaces.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.38: The Auto Create Options Dialog Box



- You can specify that all of the resulting interfaces are created using **One to One Pairing**, so that a single zone is assigned to each side of all of the mesh interfaces. Using this option may decrease your computational expense and help with troubleshooting later on, though it may produce a larger number of mesh interfaces for you to manage.

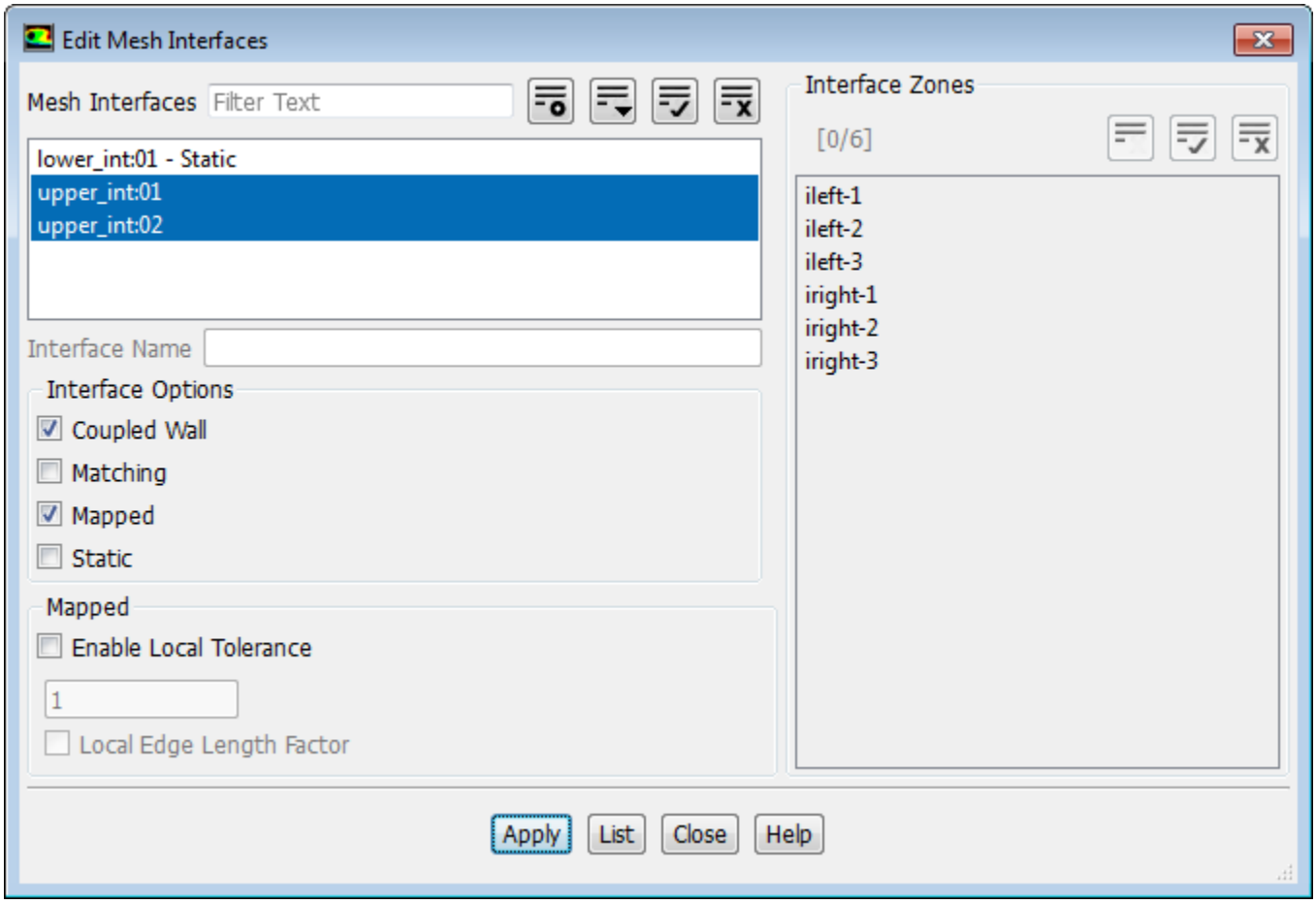
- You can specify that all of the resulting interfaces use the **Static** option, if and only if you know that the interface zones will not be moving or deforming relative to each other at the interface. While you will have the ability to enable the static option after the interfaces are created, doing so prior to creation avoids the creation of unnecessary zones (and thus saves memory usage and processing time, which can be significant when you have selected many **Unassigned Interface Zones**).

d. Click the **Auto Create** button in the **Mesh Interfaces** dialog box to create the interfaces. The resulting interfaces will be added to the **Mesh Interfaces** list. Note that when all of the unassigned zones get assigned, the **Unassigned Interface Zones** group box will become hidden, in order to simplify the user interface.

6. Make a selection from the **Mesh Interfaces** list and click **Edit...**, and then complete the setup for the mesh interfaces in the Edit Mesh Interfaces Dialog Box (p. 2696) that opens.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.39: The Edit Mesh Interfaces Dialog Box



In the **Edit Mesh Interfaces** dialog box, select the **Mesh Interfaces** you want to change, revise the settings, and click **Apply**. You can edit the **Interface Name** and/or the **Interface Zones** that comprise the interface (provided that the zones you select are unassigned and located appropriately), though such changes are only allowed if you select a single interface. You can select one or more interfaces and enable the following **Interface Options**; note that you can click the button to group the interfaces in the list by their currently enabled interface option(s), which allows you to select / deselect all those of a particular type by simply clicking the top-level branch.

- Enable **Coupled Wall** if you would like to model a thermally coupled wall between two fluid zones that share a non-conformal interface.

  Important

  If an interface has a solid zone on one or both sides, it is coupled by default. Therefore, no action is required in the **Edit Mesh Interfaces** dialog box to set up such an interface.

- Enable the **Matching** option if only internal zones should be created (that is, interior or wall / shadow pairs), since the interface zones on both sides are aligned. With the **Matching** option, even interface zones that are not perfectly aligned are treated as if they are; however, if the discrepancy between the interface zones on both sides exceeds default thresholds, then warning messages will be displayed. See Matching Option (p. 157) for more information about the recommended uses of this option.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Enable the **Mapped** option when there is penetration or large gaps between interface zones. See The Mapped Option (p. 158) for additional information.

  Enable **Enable Local Tolerance** in the **Mapped** group box for the selected interface, if you want to override the global tolerance defined for **Mapped Tolerance** in the Interface Options Dialog Box (p. 2717).

- Enable **Static** if and only if you know that the interface zones will not be moving or deforming relative to each other at the interface. This option will reduce memory usage and processing time during the running of the calculation, especially when there are many zones on both sides of the interface. The only options it is compatible with are the **Coupled Wall** and/or **Matching** options. See The Static Option (p. 160) for details.

Note the following:

- The **Periodic Boundary Condition** and **Periodic Repeats** options are not available in the **Edit Mesh Interfaces** dialog box, and so cannot be changed from enabled to disabled (or vice-versa) for an existing interface; to change them, you must delete the interface and create it again with those options revised.

- For interfaces that have the **Periodic Boundary Condition** option enabled, you can edit the **Type** and **Offset** from the **Edit Mesh Interfaces** dialog box. Note that the **Auto Compute Offset** option must be disabled to edit the **Offset**.

When all your changes are applied, close the **Edit Mesh Interfaces** dialog box.

7. The buttons at the bottom of the **Mesh Interfaces** group box allow you to take further actions, such as deleting or drawing a selected interface. Note the following:

   - The **Delete** button in the **Mesh Interfaces** dialog box can only be used on one mesh interface at a time. If you would instead like to delete multiple interfaces in a single action, you can select them in the tree and use the right-click menu command.

   - The **Options...** button opens the Interface Options Dialog Box (p. 2717), which allows you to update which mesh interfaces use the mapped option and the related settings.

8. Additional wall and interior zones will be created (and deleted) along with the mesh interfaces. These zones will be named with the same prefix as their associated interface, and are visible in the tree (for further details, see Interface Zones Automatic Naming Conventions (p. 160)). There are times when you may need to take action on these additional zones, including the following:

   - If the two interface zones did not overlap entirely, check the boundary zone type of the zone(s) created for the non-overlapping portion(s) of the boundary. If the zone type is not correct, you can use the Boundary Conditions Task Page (p. 2389) to change it.

   - If you have any **Coupled Wall** type interfaces, define boundary conditions (if relevant) by updating the interface wall zones using the **Boundary Conditions** task page.

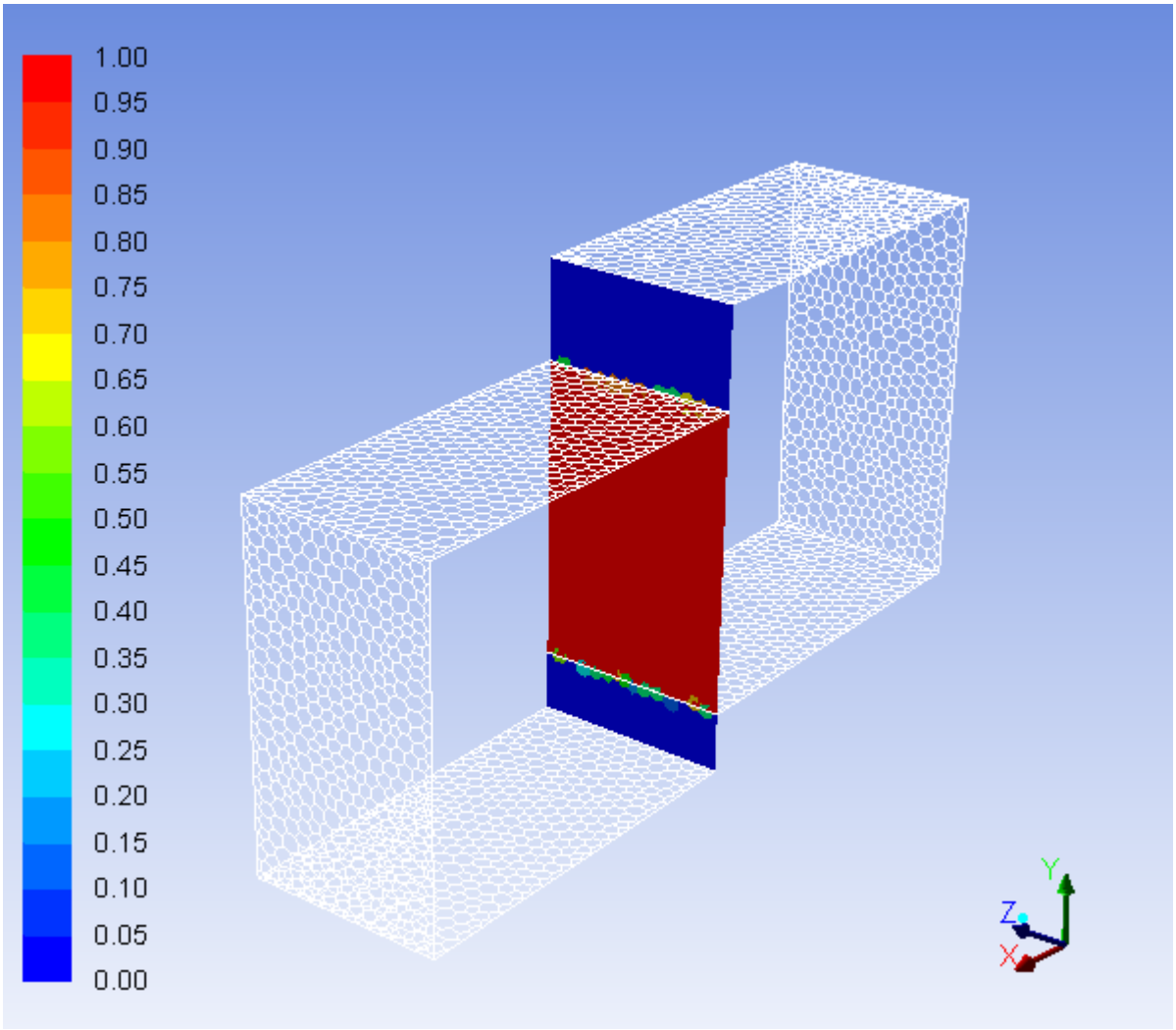
     Setup → **Boundary Conditions**

9. It is always a good practice to initialize the solution and review your interfaces:

   - You can visualize the mesh interface(s) you have created by displaying contours of **Mesh** and **Interface Overlap Fraction** in the Contours Dialog Box (p. 2676). See Figure 5.40: Contours of Interface Overlap Fraction (p. 169) for an example.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.40: Contours of Interface Overlap Fraction



- You can print information in the console about the areas of the zones associated with interfaces either by using the **List** button in the **Mesh Interfaces** dialog box or by entering the text command `define/mesh-interface/list`, as shown in the following example:

```
>define/mesh-interfaces/list
List of Mesh Interfaces


Interface Name:     int:01
Interface Options:  Coupled


Description                        Name                    ID          Area          Area Percentage
---------------------  -------------------------------  --------  ------------------  -----------------
     Interface-Zone-1                             f-1        18        0.0010104603
     Interface-Zone-2                             s-1         6        0.0010102915
      Boundary-Zone-1          int:01-side1-wall-f-1        52        0.0001105132         10.936916%
      Boundary-Zone-2          int:01-side2-wall-s-1        53        0.0001103031         10.917952%
Interface-Wall-Zone-1               int:01-wall1-1-1        51        0.0008975386         88.824722%
Interface-Wall-Zone-2        int:01-wall1-1-1-shadow        54        0.0008975386         88.839561%
```

The **Area Percentage** for **int:01-side1-wall-f-1** is calculated as the area of **int:01-side1-wall-f-1** divided by the area of **f-1**. The **Area Percentage** for **int:01-wall1-1-1-shadow** is calculated as

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the area of **int:01-wall1-1-1-shadow** divided by the area of **s-1**. The remaining area percentages are calculated in a similar manner.

Note

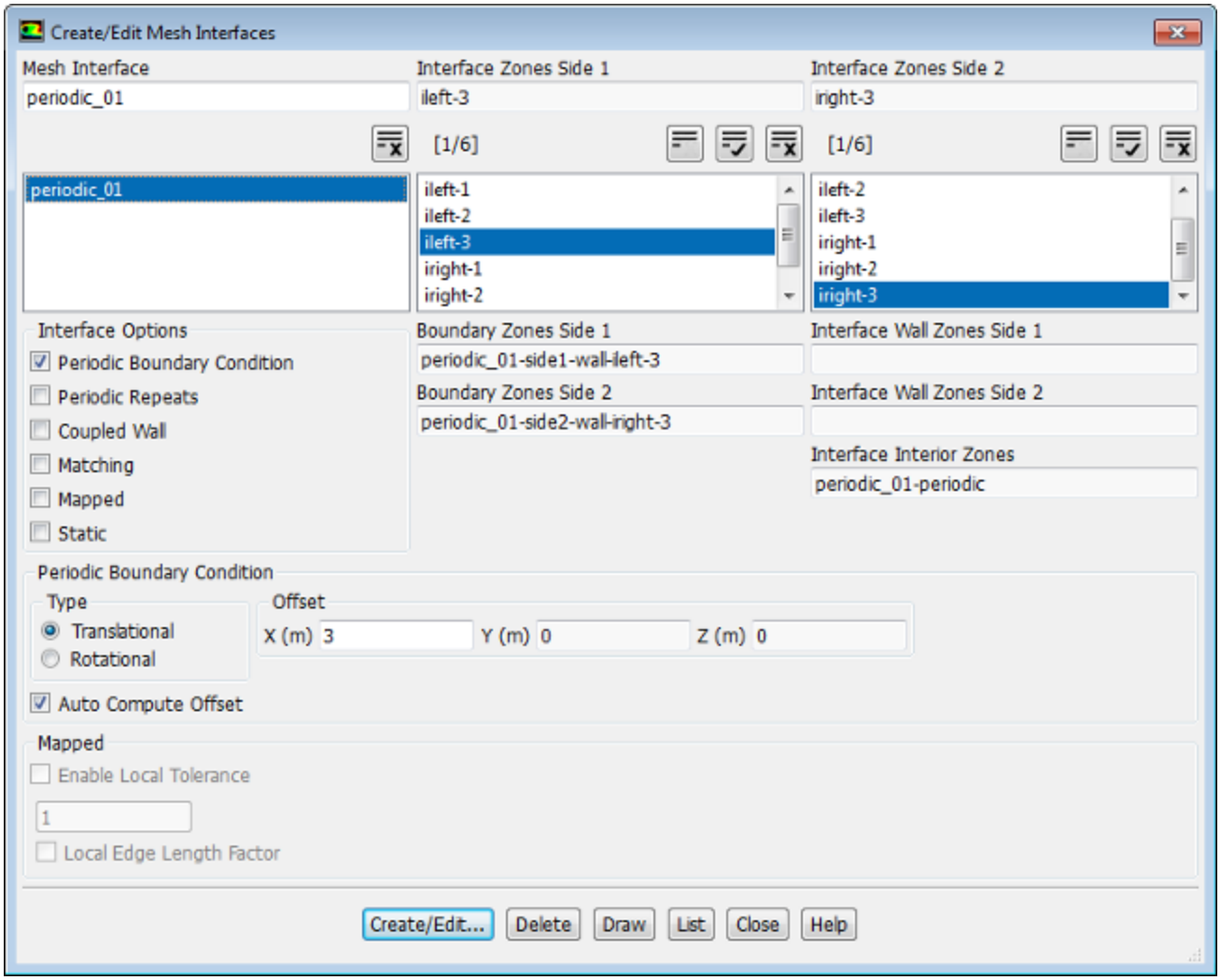
For case files in which the interface was created in ANSYS Fluent version 17.2 or earlier, you may reduce quality issues (such as left-handed faces) and/or convergence problems if you recreate the interface in version 18.0 or later by entering the following Scheme command in the console: (`recreate-sliding-interfaces`).

5.4.4.1. Manually Creating Mesh Interfaces

While it is possible to create every type of mesh interface using the manual method, it is only necessary when you want your interface to use the periodic or periodic repeats option. For all other types it is more convenient to use the automatic method (as described in the preceding section), especially when you have many interface zones and/or are unfamiliar with their names / locations.

To manually create a mesh interface, you will use the Create/Edit Mesh Interfaces Dialog Box (p. 2680), which is opened by clicking the **Manual Create...** button in the Mesh Interfaces Dialog Box (p. 2732).

Figure 5.41: The Create/Edit Mesh Interfaces Dialog Box



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Enter a name for the interface in the **Mesh Interface** text-entry box.

2. Specify the interface zones that comprise the mesh interface by selecting one or more zones in the **Interface Zones Side 1** list and one or more zones in the **Interface Zones Side 2** list. Note that you must select only one zone per side if you plan to use the periodic or periodic repeats option.

   Important

   If one of your interface zones is much smaller than the other, you should specify the smaller zone as **Interface Zones Side 1** to improve the accuracy of the intersection calculation.

3. Enable the desired **Interface Options**:

   - Enable **Periodic Boundary Condition** to create a non-conformal periodic boundary condition interface.

     – Select either **Translational** or **Rotational** as the periodic boundary condition **Type** to define the type of periodicity.

     – Retain the enabled default setting of **Auto Compute Offset** if you want ANSYS Fluent to automatically compute the offset. After creating the interface, the offsets will be displayed in these fields. The fields will be uneditable when the **Auto Compute Offset** is enabled.

     – Disable **Auto Compute Offset** if you decide that you do not want ANSYS Fluent to find the offset. In this case, you will have to provide the offset coordinates or angle in the required fields, depending on whether **Translational** or **Rotational** periodicity is selected.

     Important

     Auto computation means that the rotational angle or the translational offset will be automatically calculated and used while creating a non-conformal periodic boundary condition interface. However, it still relies on the **Rotational Axis Origin** and the **Rotational Axis Direction** that was entered for the cell zone in the cell zone condition dialog box (for example, the **Fluid** dialog box). Therefore, before proceeding with the creation of the non-conformal periodic boundary condition interface, you have to correctly enter the rotational axis for the corresponding cell zone.

     Note that auto computation of the non-conformal periodic boundary condition offset does not mean that the **Rotational Axis Origin** and **Rotational Axis Direction** are also detected automatically and updated. It is still your responsibility to set up these values correctly.

   - Enable **Periodic Repeats** when each of the two cell zones has a single pair of conformal or non-conformal periodics adjacent to the interface (see Figure 5.31: Translational Non-Conformal Interface with the Periodic Repeats Option (p. 155)). This option is typically used in conjunction with the sliding mesh model, when simulating the interface between a rotor and stator.

   - Other options are available, as described in the preceding section. Note that only **Matching** is compatible with **Periodic Boundary Condition**, and only **Coupled Wall** is compatible with **Periodic Repeats**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click **Create/Edit...** to create a new mesh interface. ANSYS Fluent will automatically create additional wall and/or interior zones for the interface, which will appear under **Boundary Zones Side 1**, **Boundary Zones Side 2**, **Interface Wall Zones Side 1**, **Interface Wall Zones Side 2**, and **Interface Interior Zones**.

5. The buttons at the bottom of the dialog box allow you to take further actions, such as the following:

   - If you create an incorrect mesh interface, you can select it in the selection list and click the **Delete** button to delete it. Any additional wall and/or interior zones that were created when the interface was created will also be deleted.

   - To edit an interface, select it in the selection list and click **Create/Edit...**, and then revise the settings using the Edit Mesh Interfaces Dialog Box (p. 2696) that opens.

   - You can click the **Draw** button to display interface zones or mesh interfaces in the graphics window. Note that you can only select and display interface zones from **Interface Zones Side 1** or **Interface Zones Side 2** if no interfaces are selected in the selection list and if the zone in question is not yet assigned to an interface. After an interface is defined, you can select the appropriate mesh interface and click the **Draw** button to display the zones under **Interface Zones Side 1** and **Interface Zones Side 2** together. Drawing is particularly useful if you want to check the location of the interface zones prior to setting up a mesh interface.

5.5. Overset Meshes

Fluent allows you to build up the computational domain from overlapping meshes—also known as Chimera or overset methodology. This gives you another way to build the complete mesh, in addition to using conformally connected cell zones, non-conformal interfaces, and mapped mesh interfaces.

5.5.1. Introduction
5.5.2. Overset Topologies
5.5.3. Overset Domain Connectivity
5.5.4. Diagnosing Overset Interface Issues
5.5.5. Overset Meshing Best Practices
5.5.6. Overset Meshing Limitations and Compatibilities
5.5.7. Setting up an Overset Interface
5.5.8. Postprocessing Overset Meshes
5.5.9. Writing and Reading Overset Files

5.5.1. Introduction

Whereas non-conformal interfaces connect cell zones along matching face zones, overset interfaces connect cell zones by interpolating cell data in the overlapping regions. For overset meshing to be successful, the cell zones must overlap sufficiently. An advantage of overset meshing is that the individual parts of an overset mesh can be generated independently and with fewer constraints than if the parts had to fit together conformally or along non-conformal interfaces. This can make it easier to replace parts of a mesh without having to remesh large parts or even the complete mesh.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.42: Overset Component and Background Mesh

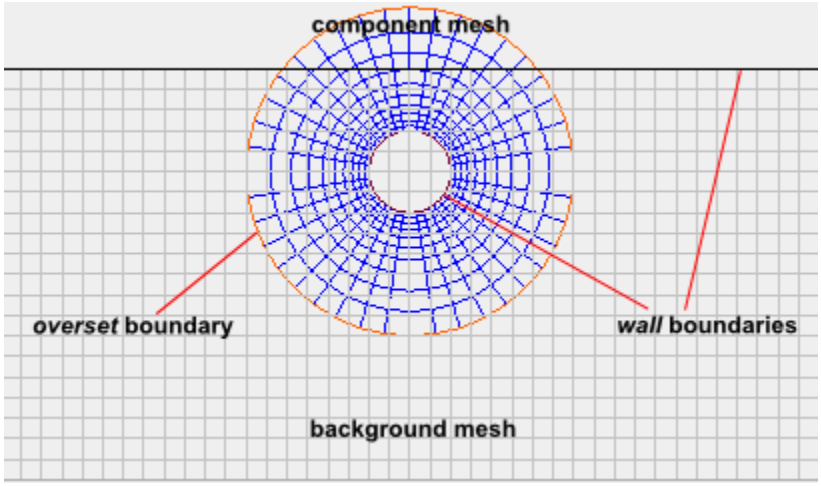
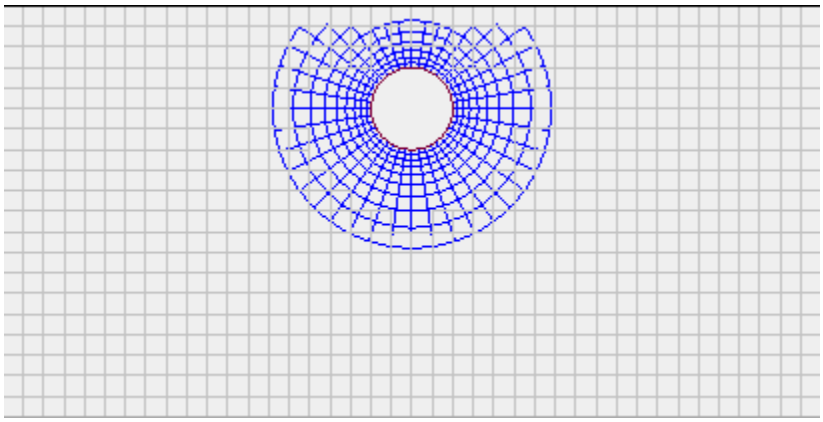


Figure 5.42: Overset Component and Background Mesh (p. 173) shows a simplified mesh for a simulation of flow over a cylinder in a duct. The mesh consists of two parts—a *background* mesh representing the duct and a separate *component* mesh around the cylinder. The case is set up in Fluent as an overset case by specifying the outer boundary of the cylinder mesh as **overset** (boundary type), and by creating an overset interface containing the two cell zones. With these two steps complete, Fluent automatically establishes the necessary connectivity between the meshes when the flow is initialized. In this process, cells that fall outside the computational domain are classified as *dead* cells. The cells where the flow equations are solved are referred to as *solve* cells. *Receptor* cells receive data interpolated from another mesh. The *donor* cells—the cells where the receptors get their data—are a subset of the solve cells. For the above case, Figure 5.43: Solve Cells Post Initialization (p. 173) shows the solve cells of the mesh after the flow is initialized. Note that in parallel, you can print partition statistics to get information about some of the cell classifications, as described in Interpreting Partition Statistics (p. 2052).

Figure 5.43: Solve Cells Post Initialization



There is no limit on the number of cell zones that can be paired in an overset interface. Fluent classifies the cell zones in an overset interface as either *background* or *component*. Background zones are the cell zones which make up the background mesh of the computational domain. They must be conformally connected and cannot have **overset** boundaries. Component zones overlay background zones and have **overset** boundaries near where they connect to the background and other component zones.

All cell types generally supported in Fluent are supported with overset meshing, including polyhedral cells in 3D. The different zones paired in an overset interface can have different element types or can have mixed element types.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Overset meshing in Fluent is compatible with mesh adaption. The mesh zones of an overset interface can be locally adapted using all available adaption tools.

Overset meshing can be used for steady-state or transient solutions. Note that for steady-state cases in parallel that use the default **Metis** partitioning, ANSYS Fluent will automatically repartition overset meshes when the solution is initialized or data is read; a model-weighted partitioning that is designed for optimal performance is used.

5.5.2. Overset Topologies

There are a multitude of configurations that can be constructed by superposing background and component meshes in an overset interface. Fluent analyzes the topology of an interface with minimal user input, which requires that the superposed meshes adhere to some topological constraints.

A case can have multiple overset interfaces, but a cell zone can only belong to a single interface. That is, interfaces cannot share component or background meshes. It is always possible to have a single overset interface for an entire model. However, it is more efficient to declare multiple interfaces if the zonal topology allows it. Only including cell zones that are absolutely required in an interface also increases efficiency. Any overset interface must have at least one background and one component mesh.

An overset interface can contain multiple component and background meshes, provided that the background meshes are conformally connected and do not intersect each other.

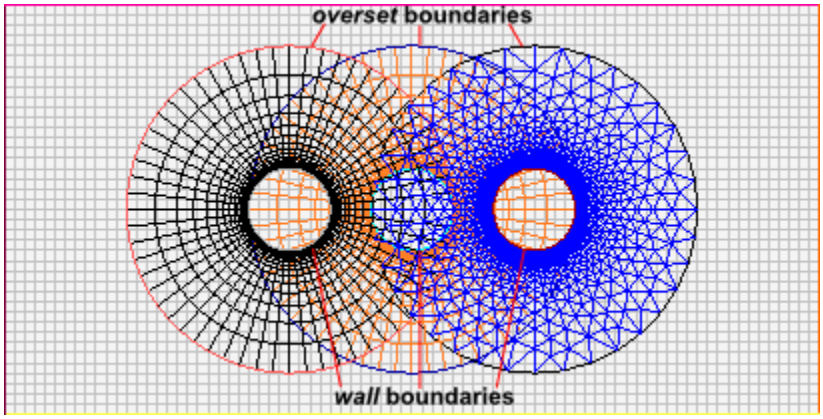
Currently, non-conformal interfaces are not fully compatible with overset meshing. Component meshes cannot be connected to a non-conformal interface and background meshes cannot have non-conformal interfaces between them if they are part of the same overset interface.

The component meshes of an overset interface are allowed to overlap arbitrarily and multiple component meshes can be combined to create an object of interest. However, an important topological constraint is that physical boundary zones (wall, inlet, outlet, symmetry, and so on) are not allowed to *intersect* each other. That is, a wall boundary cannot cross another wall boundary. For example, in the model shown in Figure 5.42: Overset Component and Background Mesh (p. 173), the component mesh cannot be placed such that its wall boundary zone intersects with the wall of the background mesh.

Overset boundaries can intersect with other overset boundaries and with physical boundaries. In Figure 5.42: Overset Component and Background Mesh (p. 173) the overset boundary of the component mesh extends outside of the intended computational domain, crossing the wall boundary of the background mesh—this is allowed. Figure 5.44: Valid Overset Meshes with Components in Close Proximity (p. 175) shows a valid configuration of three cylinders in proximity (none of the physical boundaries intersect with other physical boundaries).

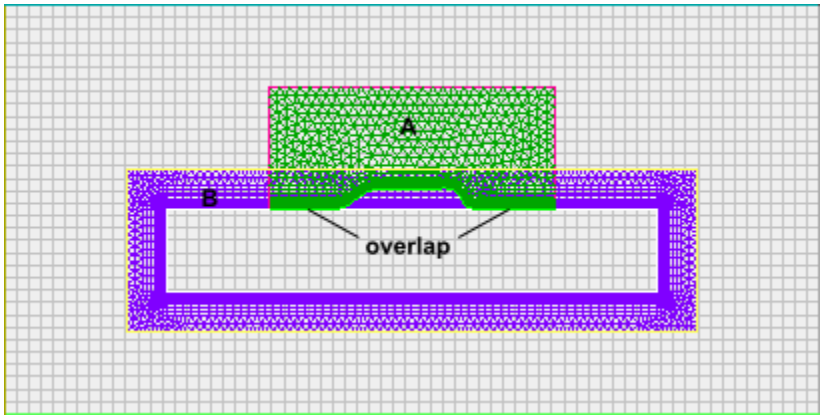
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.44: Valid Overset Meshes with Components in Close Proximity



Physical boundary zones are allowed to *overlap*, that is, a wall boundary (or parts of a wall boundary) can be coincident with another wall boundary, as long as they do not cross. This powerful functionality lets you build a body from overlapping parts or modify a body by adding parts. Figure 5.45: Second Component Modifying Existing Body (p. 175) shows an overset configuration where a second component mesh is added to an existing body to modify its shape. Fluent analyzes and automatically detects the overlapping boundaries (walls in this case) of the different meshes in an overset interface, when deciding which cells are solve cells and which cells are dead cells.

Figure 5.45: Second Component Modifying Existing Body

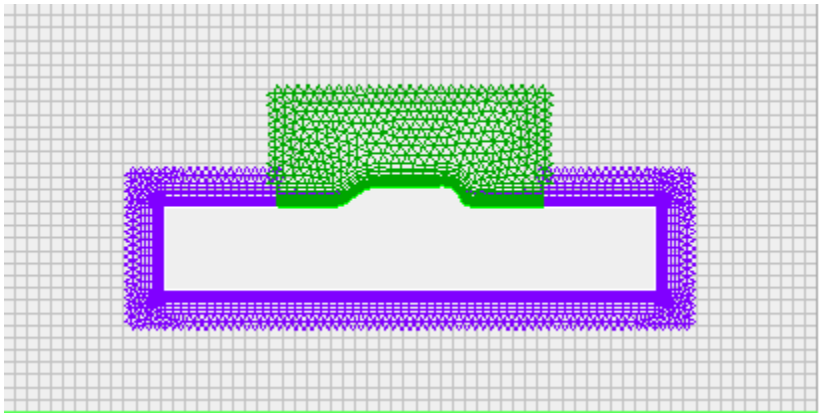


*Note that part of the wall boundary of mesh **A** lays on top of part of the wall boundary of mesh **B**, overlapping it, but the walls do not cross or intersect.*

Figure 5.46: Existing Body Modification Post Initialization (p. 176) shows the solve cells of the mesh after the flow field is initialized.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.46: Existing Body Modification Post Initialization



The previous example shows a case where physical boundaries overlap between two component meshes. Overlapping boundaries can also exist between component and background meshes.

If a body can only be built from components that have intersecting boundaries, then you must use a *collar* mesh to properly connect them. A collar mesh is an additional component mesh that bridges the intersecting physical boundaries, essentially transforming the intersection problem (not allowed) into a problem of overlapping boundaries (allowed). Figure 5.47: Multiple Components Bridged by Collars Meshes (p. 176) shows an overset interface consisting of two component meshes that have intersecting wall boundaries. Two collar meshes (labeled **C** and **D**) are manually created to correctly define the connectivity where the wall boundaries of the original component meshes (labeled **A** and **B**) intersect. The wall boundaries of meshes **C** and **D** overlap with the wall boundaries of meshes **A** and **B**.

Figure 5.47: Multiple Components Bridged by Collars Meshes

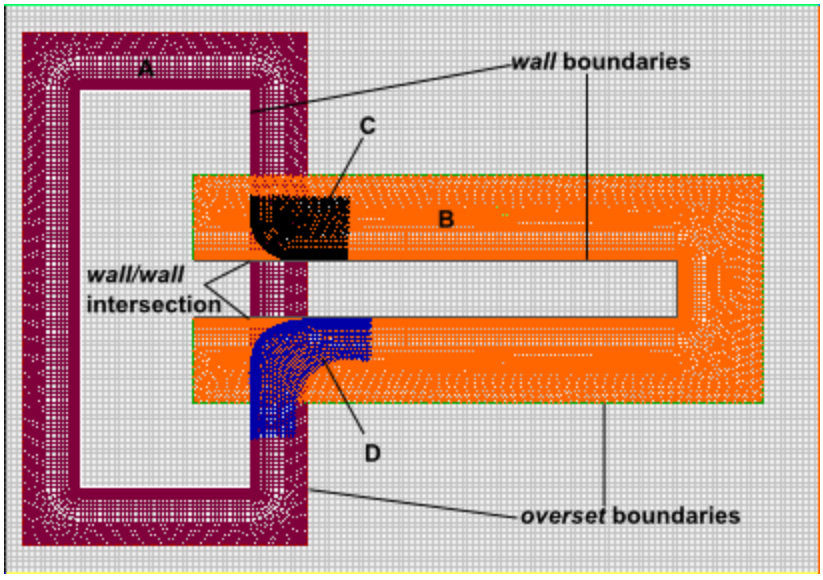
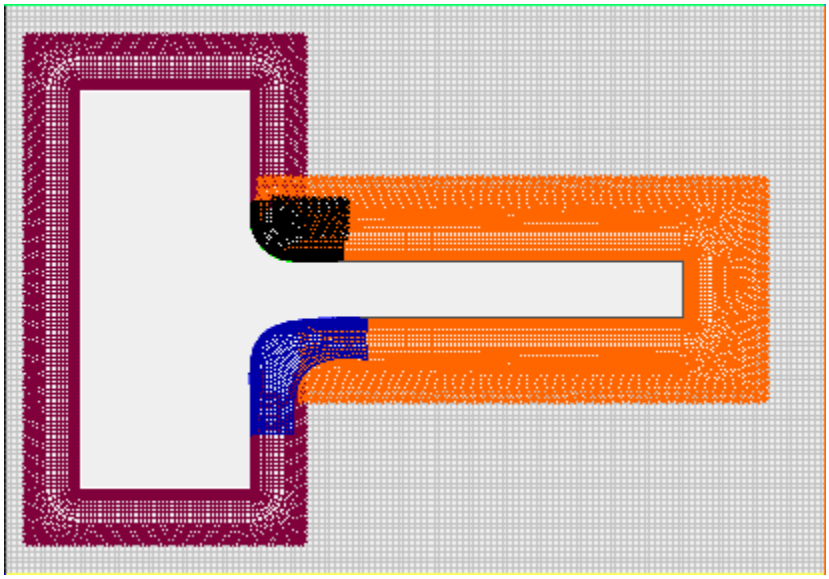


Figure 5.48: Multiple Components with Collar Meshes Initialized (p. 177) shows the resulting solve cells after the flow is initialized. The collar meshes act as fillet meshes that place the actual boundary intersections into the interior of the body and outside of the fluid domain.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.48: Multiple Components with Collar Meshes Initialized



Baffles (coupled walls) are permissible in overset meshing—both in background and component meshes.

5.5.3. Overset Domain Connectivity

When an overset interface is initialized, there are three main steps that Fluent completes to establish connectivity between the participating zones:

5.5.3.1. Hole Cutting
5.5.3.2. Overlap Minimization
5.5.3.3. Donor Search

5.5.3.1. Hole Cutting

Hole cutting is the process by which cells lying outside of the flow region (that is, inside bodies and outside of the computational domain) are marked as dead cells. This is achieved by marking all the cells that are cut by physical boundary zones (wall, inlet, outlet, symmetry, and so on), and marking seed cells determined to lie outside the flow region for subsequent flood filling of dead cells. The result of this flood filling is a valid overset mesh with maximum mesh overlap.

Figure 5.49: Overset Component and Background Meshes Before Hole Cutting (p. 178) and Figure 5.50: Overset Component and Background Meshes After Hole Cutting (p. 178) show an overset mesh before and after hole cutting. All of the background cells that are bounded or cut by the cylinder wall are classified as dead during hole cutting. The maximum overlap mesh shown in Figure 5.50: Overset Component and Background Meshes After Hole Cutting (p. 178) is a valid overset mesh, however, it may not be ideal. An overly large overlap between component and background meshes is computationally inefficient (the equations are solved in more cells than are necessary). Additionally, the cell sizes of the overlapping meshes may vary greatly—this affects the data interpolation and is detrimental to solution quality. Ideally meshes should transition in regions of similar resolution.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.49: Overset Component and Background Meshes Before Hole Cutting

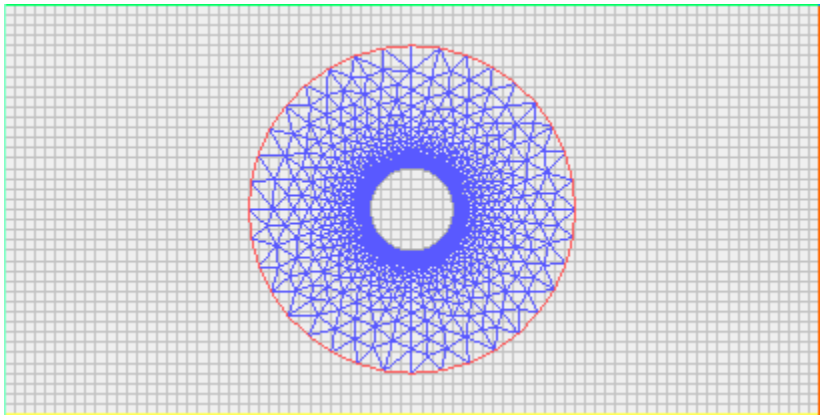
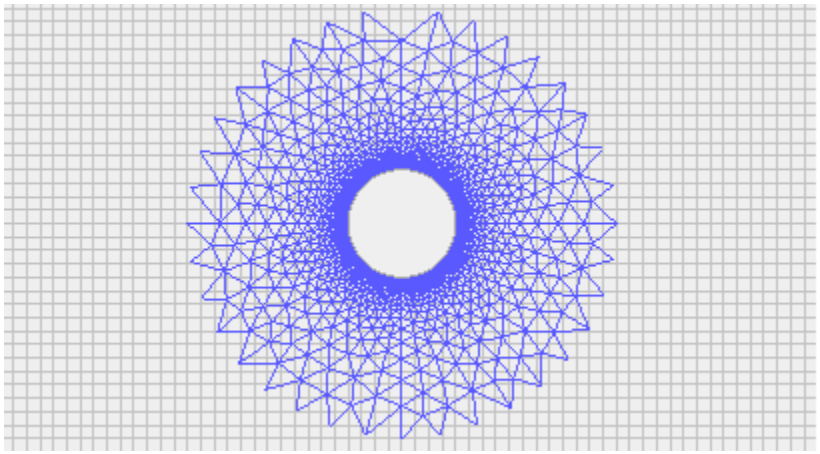


Figure 5.50: Overset Component and Background Meshes After Hole Cutting

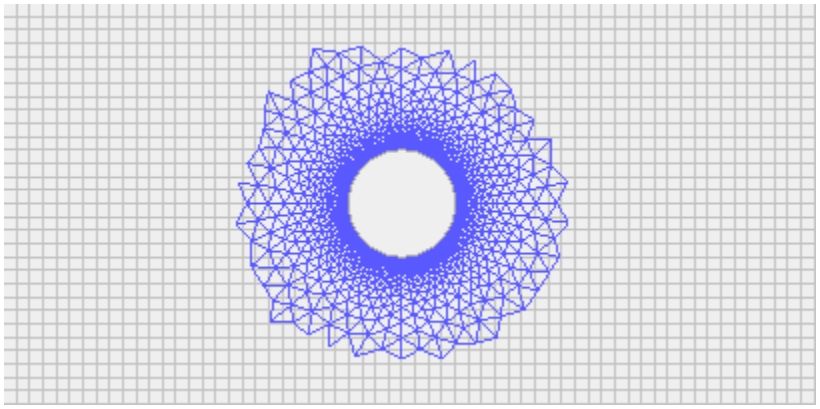


5.5.3.2. Overlap Minimization

Overlap minimization is used to minimize mesh overlap among different component and background meshes by converting additional solve cells into receptor cells and turning unnecessary receptors into dead cells. During this process, a solve cell is turned into a receptor cell if the cell can find a suitable donor cell with higher donor priority. By default, smaller cells have a higher donor priority. Thus, in mesh overlap areas, without additional user input the solver attempts to obtain the solution on the finest local mesh. The resulting mesh interface moves to an area where the meshes are more comparable in cell size, leading to better solution quality. Figure 5.51: Overset Component and Background Meshes After Overlap Minimization (p. 179) shows the cylinder case from the previous figure after the overlap minimization step.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.51: Overset Component and Background Meshes After Overlap Minimization



Note that overlap minimization is not literally minimizing the mesh overlap. The overlap minimization is based on a flood filling of receptor cells, wherever suitable solve cells with higher donor priority can be found. The resulting actual mesh overlap depends on the local donor priority distribution.

Also note that with overlap minimization, data interpolation between cell zones does not necessarily occur at the overset boundaries. The purpose of specifying an overset boundary is primarily to specify that overset mesh coupling should happen, and not where it should occur.

You have the option to specify different donor priority methods in order to control the location of the overset interface. The text user interface command `define/overset-interfaces/options/donor-priority-method` allows you to specify that the donor priority is based either on the cell size (proportional to the inverse of the cell volume) or the boundary distance (proportional to the inverse of the distance to the closest boundary). Overlap minimization with donor priority based on cell size (default) works best if the component mesh resolution is fine near walls and increases away from walls, to spacings similar to—or larger than—the background mesh (see Figure 5.49: Overset Component and Background Meshes Before Hole Cutting (p. 178) for an example of this type of mesh). If the overlapping meshes have uniform and nearly identical resolutions, then overlap minimization based on cell size will not reduce the overlap and a donor priority based on the boundary distance should be used instead. The following two figures illustrate an overset mesh before and after initialization where a donor priority based on boundary distance is used. Note that for the evaluation of donor priorities based on boundary distance, all boundaries (including inlets, outlets, and so on) are used for the distance evaluation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.52: Overset Mesh Before Hole Cutting

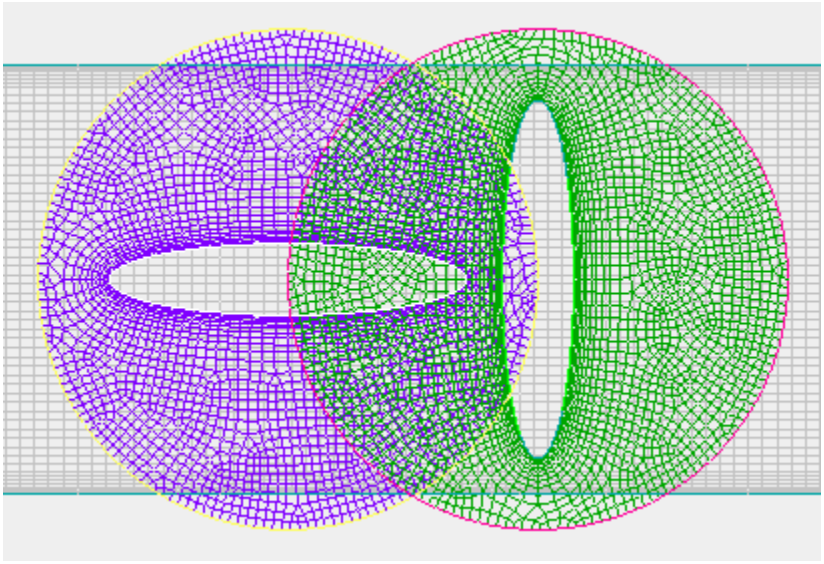
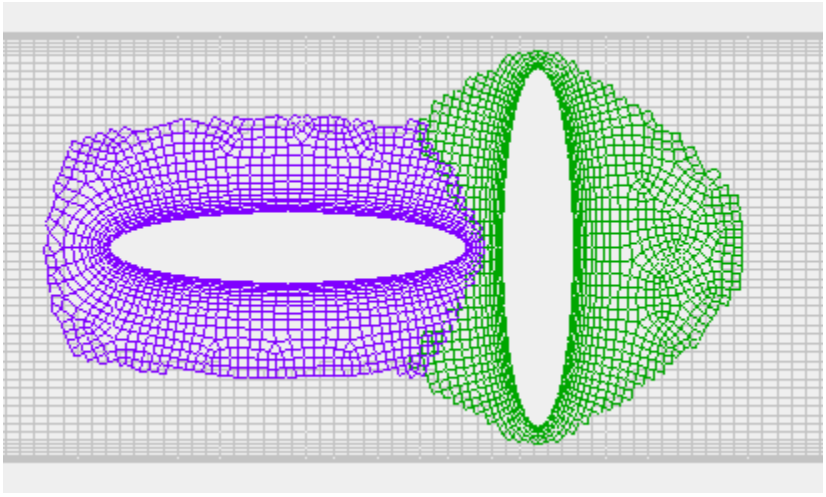


Figure 5.53: Overset Mesh After Minimization Based on Boundary Distance



You have the option for additional control over the overlap minimization process by assigning grid priorities to component and background meshes, using the text user interface command `define/overset-interfaces/grid-priorities`. Meshes (that is, cell zones) with higher grid priorities are favored during the minimization process, irrespective of the local cell donor priority. If two cell zones have the same grid priority, then the cell donor priorities are used for the overlap minimization. For example, by assigning a higher grid priority to a coarser mesh, you can force Fluent to minimize the finer mesh and obtain the solution on the coarser mesh, even if the donor priority is based on cell size. Using grid priorities, similar to using donor priority based on boundary distance, is helpful if local mesh distributions are irregular, and produce overlap regions that are irregular as well. Note that the grid priorities take precedence over the donor priorities.

You have the option of disabling overlap minimization by using the text command `define/overset-interfaces/options/minimize-overlap?`. This command is only available globally and applies to all overset interfaces in the domain.

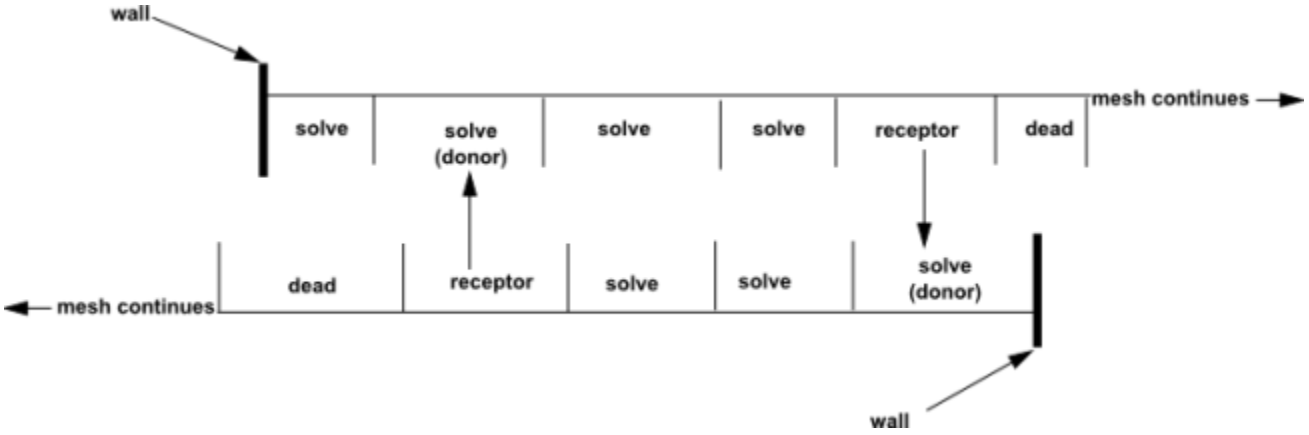
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.5.3.3. Donor Search

The donor search is the final step in establishing the domain connectivity. Fluent searches other meshes for valid solve cells for each receptor. The solve cell containing the cell centroid of the receptor cell, along with its connected solve cells, are used as donor candidates for a given receptor. Each receptor must have at least one valid donor cell.

There must be four or more cells in the overlap of both meshes to ensure a successful donor search. The receptor cells, which form the fringe layer of a mesh zone, must overlap sufficiently with the opposite mesh, such that they find valid solve cells as donors. For an example of valid mesh overlap, see Figure 5.54: Valid Overlap (p. 181).

Figure 5.54: Valid Overlap



5.5.4. Diagnosing Overset Interface Issues

Topology detection of overset interfaces in Fluent is automatic and does not require user input. However, for it to work well, it requires that the provided meshes are of sufficient quality, particularly when dealing with meshes that have overlapping boundaries.

Overset meshes are built from parts that can be meshed independently, which can lead people to think of this as a much simpler process for mesh generation. This is not always the case, as considerable thought is often required to generate good meshes for the individual components to avoid problems during overset interface initialization.

If you are faced with issues during the initialization of an overset interface, there is most often no recovery without making changes to the mesh. You must analyze the cause of the issue, using the available postprocessing and diagnostic tools, and improve the mesh before the simulation can continue.

When diagnosing overset interface failures, it can be convenient to use the text commands `define/overset-interfaces/intersect` and `define/overset-interfaces/clear` (available after enabling the expert tools available under `define/overset-interfaces/options/expert`) to execute and clear the hole cutting without having to initialize the solution.

The issues most often encountered are:

5.5.4.1. Flood Filling Fails During Hole Cutting
5.5.4.2. Donor Search Fails Due to Orphan Cells

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.5.4.1. Flood Filling Fails During Hole Cutting

These are the issues that can cause problems with flood filling:
5.5.4.1.1. Incorrect Seed Cells
5.5.4.1.2. Leakage Between Overlapping Boundaries

5.5.4.1.1. Incorrect Seed Cells

Flood filling of dead cells will fail if Fluent inadvertently identifies seed cells inside of the fluid zone instead of outside. This can occur as a result of complexities in the topology or geometry of the boundary zones, or because of mesh resolution issues. It manifests itself as an entire fluid region, or a complete cell zone, being marked as dead.

This issue can be diagnosed by using the list command with the overset verbosity set to `1` (see Overset Interface listing (p. 191)).

You can use the TUI command `define/overset-interfaces/debug-hole-cut` (available after enabling the expert tools available under `define/overset-interfaces/options/expert`) to identify the problematic seed cells. This command allows you to execute the hole cutting process on a specific overset interface and mark the seed cells. When the flood fill option of the debug command is disabled, it will create two adaption registers with dead cells and seed cells (see Displaying Registers (p. 1704)). The dead marks show the cells cut by the physical boundary zones. The seed cells (mark0 adaption register) show the cells that will be used for flood filling. It is critical that all the seed cells are located inside bodies or outside of the computational domain. If there is a seed cell in the fluid region, it will cause a flood fill failure.

It is not possible to recover from flood filling failure without making changes to the model.

5.5.4.1.2. Leakage Between Overlapping Boundaries

If no bad seed cells are found, as described in Incorrect Seed Cells (p. 182), and there are overlapping boundaries, it is likely that the overlapping boundaries do not match sufficiently and the flood filling of dead cells leaks from the inside of the body to the outside. As a result, a complete cell zone can disappear, due to having only dead cells. The permissible tolerance between overlapping boundary zones is at most one cell-size in the wall normal direction. This can be restrictive when dealing with prismatic boundary layer meshes with high cell aspect ratios at the wall.

The `define/overset-interfaces/debug-hole-cut` text command, with the flood filling option enabled, marks the cells that will be converted to dead cells (as a result of flood filling) and generates a mark1 adaption register containing those cells. If your model suffers from this leakage, you must improve the quality of the mesh, by making the overlapping boundary zones match better.

5.5.4.2. Donor Search Fails Due to Orphan Cells

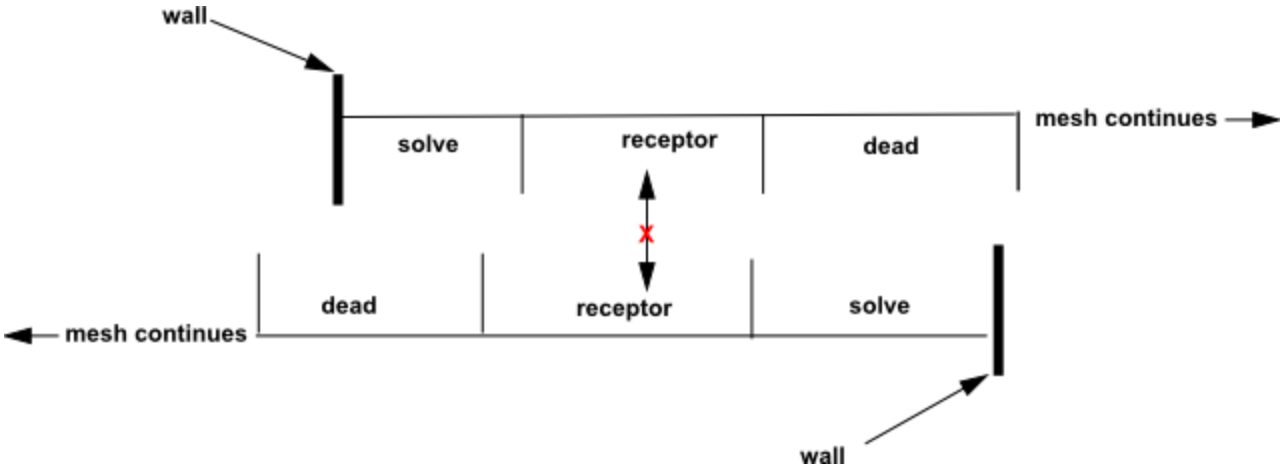
If a receptor cell cannot find a valid donor cell, then it becomes an *orphan* cell. The presence of orphan cells generally indicates that there is insufficient overlap between meshes or that the mesh resolutions do not match well. As a result, the receptors are either finding no candidate donor cells, or invalid candidates (dead or receptor cells).

Orphan cells can be an issue when walls of different meshes are in proximity. You must have at least four cells in a gap, such as the ones between the walls in Figure 5.44: Valid Overset Meshes with Components in Close Proximity (p. 175), in order to have a good connection between the meshes—otherwise orphan cells are created. When a receptor overlays another receptor, it becomes an orphan.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See Figure 5.55: Invalid Overlap Creating Orphans (p. 183) for an example of insufficient overlap causing the creation of orphan cells. For information on visualizing orphan cells, see Postprocessing Overset Meshes (p. 187). Figure 5.54: Valid Overlap (p. 181) shows meshes overlapping sufficiently to avoid the creation of orphan cells.

Figure 5.55: Invalid Overlap Creating Orphans



You can mark the orphan cells, as described in Overset Cell Marks (p. 191), which will likely indicate where the mesh overlap must be increased. Sometimes, increasing the mesh resolution can lead to a sufficient increase in mesh overlap. If mesh resolution is the issue, you can also use local mesh adaption, along with mesh modification, to refine the mesh. Most often the mesh requiring adaption is the mesh where the orphan should find its donors, and not the mesh zone containing the orphan.

If the presence of orphan cells in the simulation is unavoidable and you proceed with a calculation, by default the solver applies a numerical treatment that attempts to assign reasonable values to the orphan cells. This treatment has been found to prevent divergence in some instances, allowing the solution to proceed; however, the orphan cell treatment does not guarantee the solution quality and robustness a priori. The success of this treatment is case dependent, and difficult to predict based on orphan cell counts. It is therefore highly recommended that you make every effort to eliminate orphan cells by modifying the mesh, as described previously.

5.5.5. Overset Meshing Best Practices

Receptor and donor cell sizes should be comparable to ensure that interpolation errors are minimized. You can manually adapt the mesh to reduce the mismatch between donors and receptors.

You must have a minimum of four cells in a gap to create a valid overset interface, but a higher number is recommended to ensure robust coupling between meshes and to prevent the creation of orphan cells (see Figure 5.55: Invalid Overlap Creating Orphans (p. 183) and Figure 5.54: Valid Overlap (p. 181)).

It is recommended that you start transient simulations from a converged steady-state solution.

If you are experiencing startup issues with a steady-state case, it is recommended that you ramp-up to the final boundary conditions. For example, you can begin with a lower inlet velocity and a higher viscosity to establish flow, before increasing to the target velocity and the desired final material properties.

When using automatic timestep calculation for the pseudo-transient method, it is recommended that you use the user-defined length scale option.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you replace a zone that is part of an overset interface, you should either reinitialize the solution or patch the solution on the replaced zone(s) before continuing with the calculation. Alternatively, you can disable **Interpolate Data** in the **Replace Cell Zone** dialog box when you are replacing the zone(s).

It is recommended that you use the double-precision solver for overset mesh simulations.

Closed domain cases are defined as setups that do not have a flow inlet or outlet boundary; this type of setup is not supported with overset meshing. If such a setup is unavoidable, creating a small pressure-outlet boundary far away from the region of interest is recommended before attempting to run the case using overset meshes.

For dynamic and sliding mesh cases, note the following:

- The ideal time step size should be chosen such that the relative mesh motion does not exceed the length of the smallest cell at the overset interface for a given time step. The mesh length used for calculating the time step should be restricted to cells in the overset overlap region. This ensures a more gradual evolution of the overset interface with the motion and is recommended for better accuracy and stability.
- In general, the motion of the overset interface may change the status for individual cells from dead to receptor or solve, and vice versa. The exact instances depend on the relative mesh sizes of the moving cell zones participating in the overset interface, the magnitude of the relative mesh velocity, and the time step size. There is currently no mechanism to eliminate this type of cell status change for general cases.
- While the solver is capable of handling the cell status change, it is recommended that you actively monitor these instances by setting the overset mesh verbosity to 1 using the `define/overset-interface/options/verbosity` text command. With such a verbosity, the solver reports the instances of dead to solve cells in the console. Keeping these to a minimum is highly recommended for better accuracy and stability.
- Do not have large variations in mesh resolution in the motion path that could trigger large changes in the overset interface as the motion proceeds. The use of grid priorities can minimize large variations in the overset interface.

You can specify that the overset interfaces use either the inverse distance interpolation method (the default) or the least squares interpolation method by using the following text command. The least squares method is recommended for sliding mesh cases.

`solve` → `set` → `overset` → `interpolation-method`

Important

The least squares method is computationally more expensive than the inverse distance method.

5.5.6. Overset Meshing Limitations and Compatibilities

Overset meshing has the following limitations and compatibilities:

5.5.6.1. Limitations

Overset meshing currently has the following limitations:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Overset interfaces cannot contain **solid** cell zones.
- Component meshes cannot be connected to a non-conformal interface.
- Background meshes cannot have non-conformal interfaces between them if they are part of the same overset interface.
- Component zones cannot have **periodic** boundary conditions.
- Background zones cannot have **overset** boundaries.
- Component mesh boundaries cannot overlap with coupled walls.
- Overset meshing is not compatible with dynamic mesh remeshing or layering.
- FMG initialization is not available.
- The FAS multigrid scheme is not used.
- Implicit residual smoothing cannot be enabled.
- Overset meshing is not supported for closed domains.
- The use of node weights for node-based gradients in postprocessing is not available for cases with overset meshes.

5.5.6.2. Compatibilities

Overset meshing is compatible with the transient and steady-state solvers. Both laminar and turbulent flow regimes with heat transfer are allowed, as well as, non-porous, single phase and Volume of Fluid (VOF) multiphase flows. These applications are valid with overset for the listed solver options and configurations. Unless otherwise specified, consider all other solver features and models to be unsupported.

Supported Options and Models:

- Pressure-based solver using:
  - 2D (planar and axisymmetric) and 3D flows in the absolute velocity formulation
  - Cell-based gradient methods, Green-Gauss, and Least-Squares
  - All pressure interpolation schemes
  - First-order and second-order spatial schemes
  - Coupled scheme for pressure-velocity coupling
  - Pseudo transient method
  - First-order and (for stationary meshes only) second-order and bounded second-order transient formulations
- Density-based solver using:
  - 2D (planar and axisymmetric) and 3D flows in the absolute velocity formulation

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - Compressible flows
  - Cell-based gradient methods, Green-Gauss, and Least-Squares
  - All convective flux types (Roe-FDS, AUSM, and Low Diffusion Roe-FDS)
  - First-order and second-order spatial schemes
  - Pseudo transient method
  - First-order and (for stationary meshes only) second-order and bounded second-order transient formulations
- k-ε and standard k-ω turbulence models
- Dynamic and sliding meshes with the first-order transient formulation
- All AMG options
- All boundary conditions are supported with the following limitations:
  - Non-interior face zone boundary conditions—exhaust fan, inlet vent, intake fan, outlet vent—are only allowed on cell zones that are not participating in an overset interface
  - Interior face zone boundary conditions—fan, porous-jump, and radiator—are only allowed on cell zones that are not participating in an overset interface
- Cell zone conditions
  - Source terms may only be defined on cell zones not participating in an overset interface
  - Fixed values may only be defined on cell zones not participating in an overset interface
- Hybrid and standard initialization
- An overset meshing solution may be obtained on any mesh type, including manually adapted meshes

5.5.7. Setting up an Overset Interface

Once you have read all of the related meshes into Fluent, use the following steps to setup an overset interface.

1. Change the boundary zones of component meshes to **overset** (boundary type), where needed.

   Important

   You must correctly specify the **overset** boundaries for all zones in an overset interface before creating the overset interface. (Once overset interfaces are created you cannot change any boundaries from **overset** to another type or from another type to **overset**.)

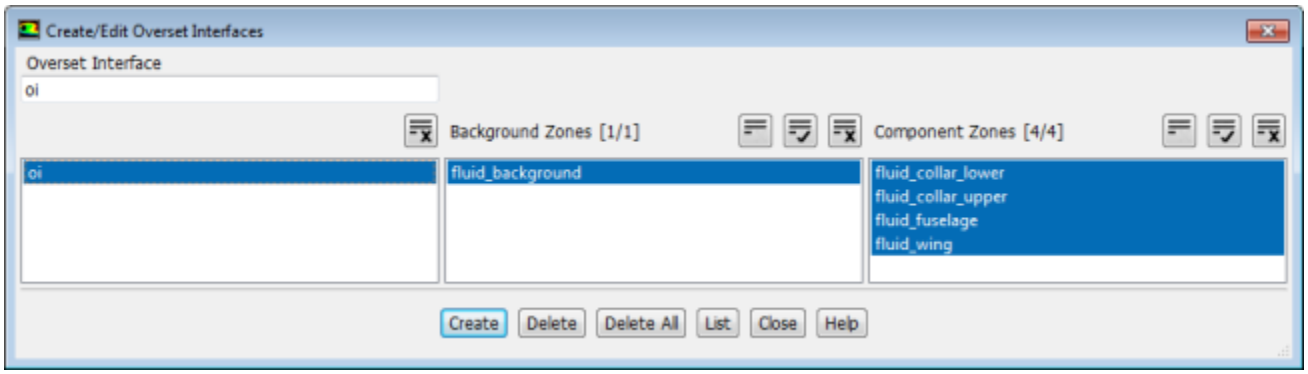
   Setup → **Boundary Conditions** → **<boundary>** → **Type** → **overset**

2. Create an overset interface between the cell zones that must be coupled. A case can have multiple interfaces. Each interface must have at least one background and one component mesh.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Up Domain → Interfaces → Overset

Figure 5.56: Create/Edit Overset Interfaces Dialog Box



a. Enter a name under **Overset Interface**.

b. Select the zones of the interface in the **Background Zones** and **Component Zones** selection lists.

c. Click **Create**.

3. [Optional]—Use the text user interface for additional control over the overlap minimization of the overset interface. It can be helpful to revise the default settings if your overlapping meshes have uniform and nearly identical resolutions or if local mesh distributions are irregular, as this may improve the computational efficiency and/or the solution quality. See Overlap Minimization (p. 178) for details.

   - To specify whether the cell donor priorities used during minimization are based on the cell size or distance to the nearest boundary, use the following text command:

     `define → overset-interfaces → options → donor-priority-method`

   - To revise the grid priorities during minimization, use the following text command. The grid priority for each zone is an integer, and only the relative values between the zones matter.

     `define → overset-interfaces → grid-priorities`

   - To disable overlap minimization entirely, use the following text command. This command is only available globally and applies to all overset interfaces in the domain.

     `define → overset-interfaces → options → minimize-overlap?`

Note that these steps only define an overset interface. Hole cutting occurs during solution initialization.

If a case contains **overset** face zones, but no defined overset interfaces, then Fluent creates a default interface with the name `default-overset-interface` upon initialization. This default overset interface contains all cell zones in the domain. If you know that some cell zones do not need to be included in an overset interface, then you can increase the solution performance by manually creating the interface and only including the relevant background and component meshes.

5.5.8. Postprocessing Overset Meshes

Fluent has several overset-specific postprocessing tools to help you analyze/understand the domain connectivity. These tools are only available after the flow field is initialized and the domain connectivity is established.

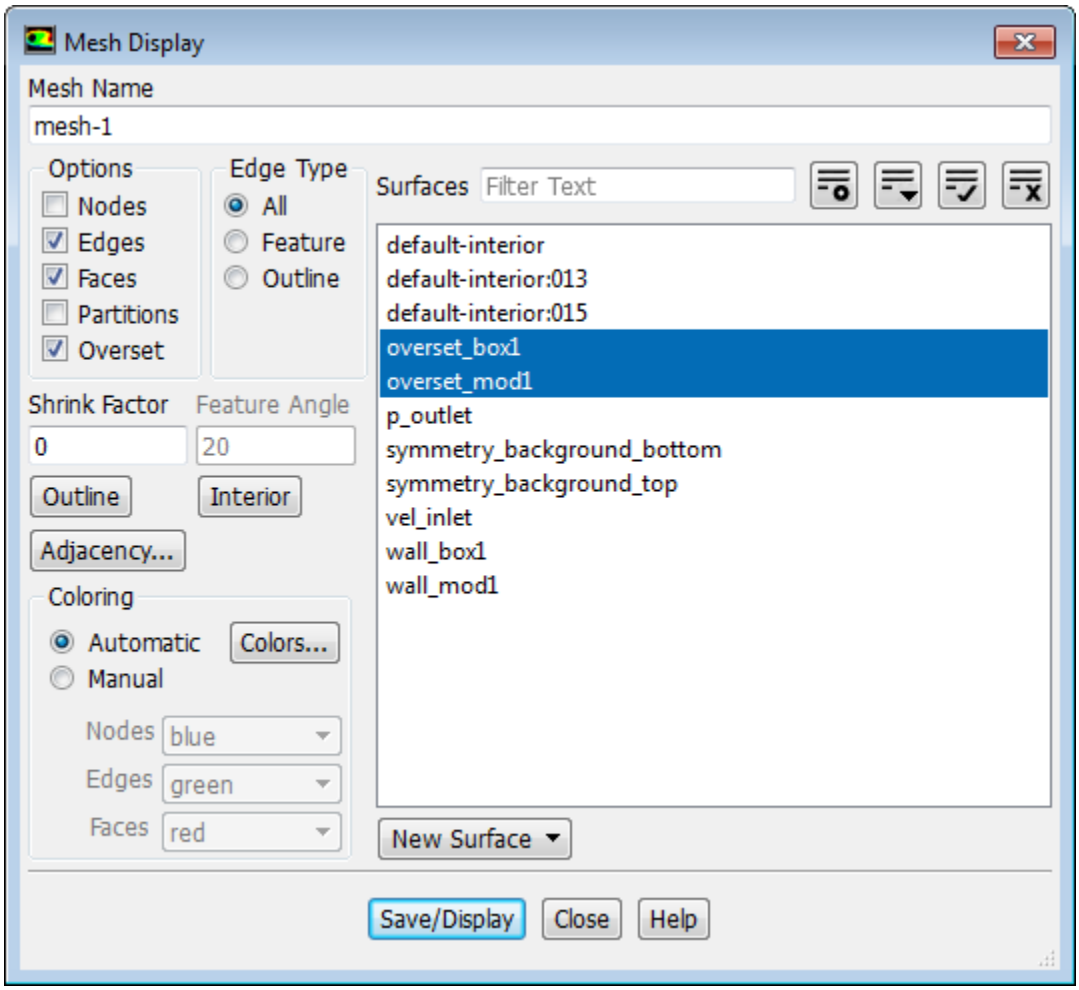
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.5.8.1. Overset Mesh Display
5.5.8.2. Overset Field Functions
5.5.8.3. Overset Cell Marks
5.5.8.4. Overset Interface listing
5.5.8.5. Overset Postprocessing Limitations

5.5.8.1. Overset Mesh Display

You can either visualize the complete mesh or just the active cells. If the **Overset** option is enabled in the **Mesh Display** dialog box, by default only solve cells are shown. Disabling the **Overset** option displays the complete surfaces.

Setting Up Domain → Mesh → Display



You have the option to have receptor cells displayed along with the solve cells when **Overset** is selected. You can enable this option via the TUI command `define/overset-interfaces/options/render-receptor-cells?`.

Note

When displaying contours of postprocessing quantities, dead cell values are never shown, regardless of whether or not **Overset** is enabled in the **Mesh Display** dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

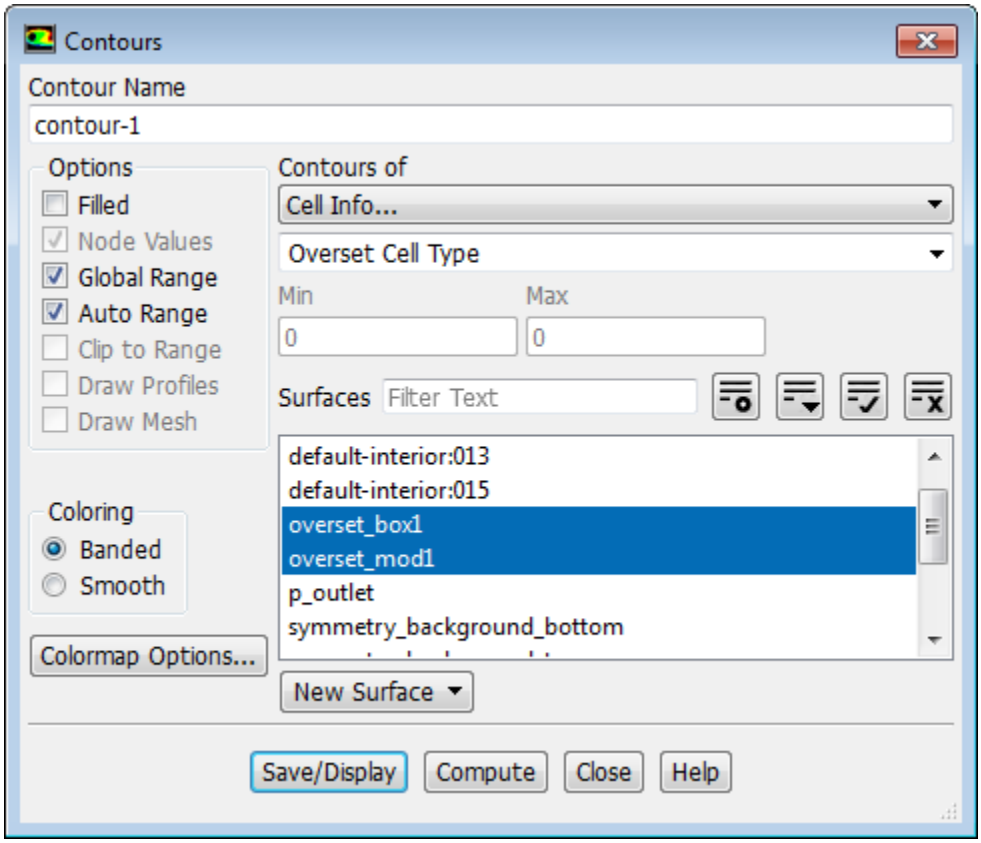
5.5.8.2. Overset Field Functions

There is an **Overset Cell Type** function available in the **Cell Info...** category in the Contours Dialog Box (p. 2676). The integer function value depends on the overset cell type:

Cell Type	Integer Function Value
donor	2
solve	1
receptor	0
orphan	–1
dead	–2

This field function is also available in the **Contours** dialog box.

Postprocessing → Graphics → Contours → Edit...



1. Select **Cell Info...** and **Overset Cell Type** from the **Contours of** drop-down lists.
2. Choose the surfaces containing the overset cells you want to visualize and click **Display**.

Figure 5.57: Contours of Overset Cell Type: Background Mesh

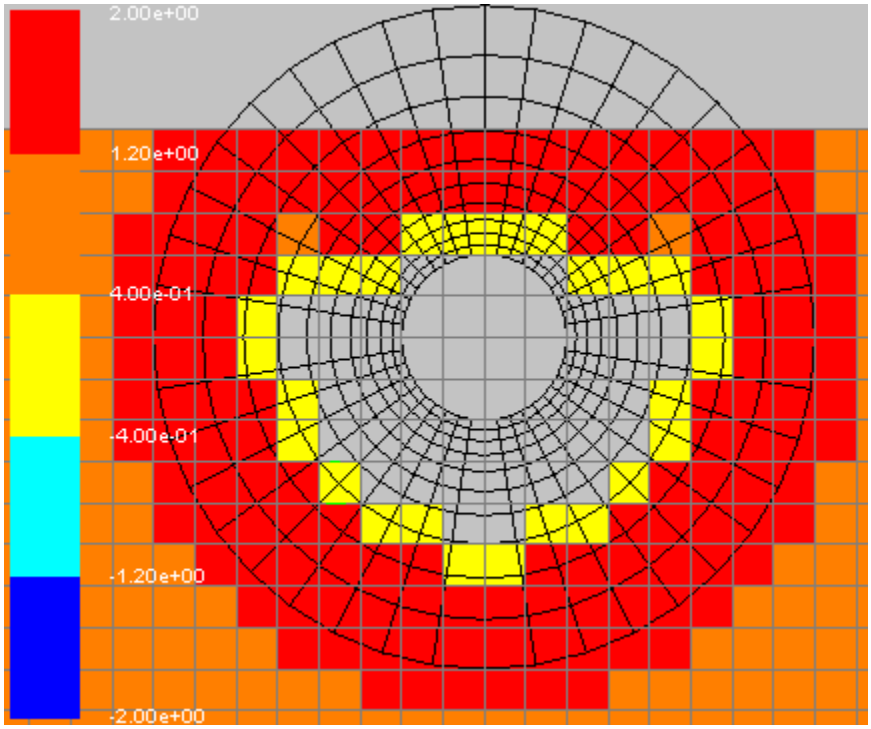


Figure 5.58: Contours of Overset Cell Type: Component Mesh

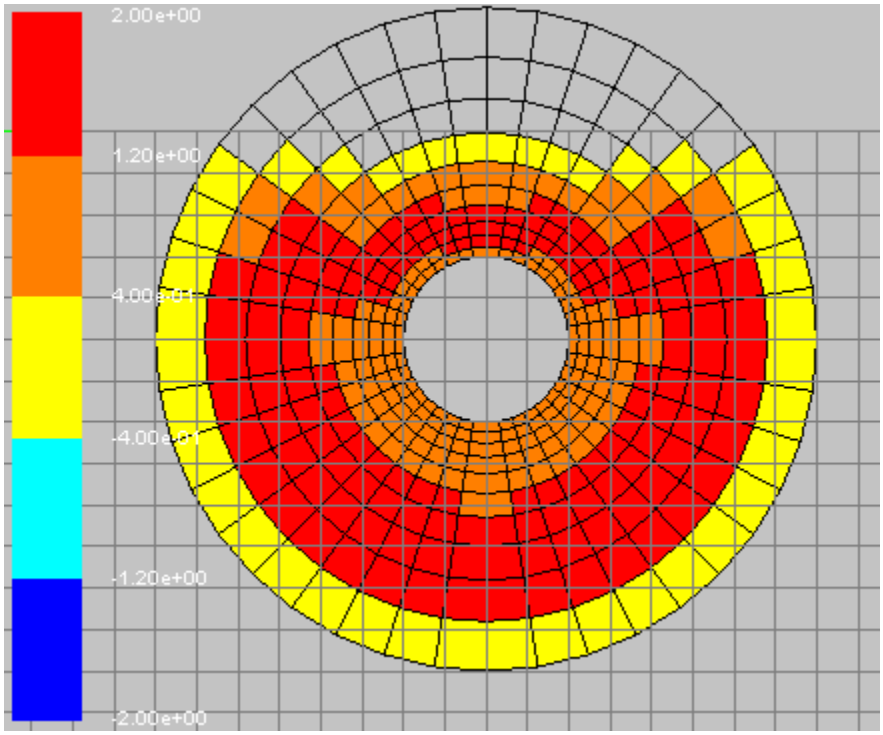


Figure 5.57: Contours of Overset Cell Type: Background Mesh (p. 190) and Figure 5.58: Contours of Overset Cell Type: Component Mesh (p. 190) display a background mesh and component mesh (sequen-

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tially) of **Overset Cell Type** for the cylinder problem shown in Figure 5.42: Overset Component and Background Mesh (p. 173).

> **Note**
>
> If you want to display orphan cells, then you must enable the TUI command `define/overset-interfaces/options/render-receptor-cells?`.

If the expert options are enabled under `define/overset-interfaces/options/expert`, then you can also choose to display **Overset Donor Count** and **Overset Receptor Count** under **Cell Info...**. The **Overset Donor Count** is filled for each receptor and shows the number of donors. Similarly, the **Overset Receptor Count** is filled for donor cells and shows the number of receptors that are referencing the donors.

Note that the donor counts reported are the maximum number of donors available for the data interpolation. The actual number of donors used by the solver may be less.

5.5.8.3. Overset Cell Marks

It can be difficult to interpret contours of overset cells in 3D. Often, it is more illustrative to mark a certain type of overset cells and display these marked cells. Fluent uses adaption registers and the **Manage Adaption Registers** functionality (see Manipulating Adaption Registers (p. 1701)) to visualize overset-specific cell markings.

Using the text command `define/overset-interfaces/mark-cells` you can mark cells of specified overset cell type (solve, receptor, donor, orphan, or dead), either globally or on a single cell zone. This command automatically fills registers that can be displayed using the **Manage Adaption Registers** functionality.

5.5.8.4. Overset Interface listing

Use the text command `define/overset-interfaces/list` to print overset interface related information to the console. If the overset verbosity, `define/overset-interfaces/options/verbosity`, is set to `1`, then counts of each overset cell type are printed per cell zone as well.

5.5.8.5. Overset Postprocessing Limitations

Volume integrals are taken over all solve cells in the domain. This results in double counting where solve cells overlap, leading to errors in volume integrals on overset meshes.

Similarly, surface integrals, including force and moment reports, have errors if the integral is taken over boundary surfaces that overlap. If there are no overlapping boundaries, surface integrals are exact.

> **Important**
>
> Errors in integral evaluations scale with the size of the overlap.

5.5.9. Writing and Reading Overset Files

When writing overset case files, it is recommended that you use the hierarchical data format (HDF), as in this format the overset domain connectivity is saved in the case file. This makes it unnecessary to

reestablish the domain connectivity when reading an overset case into a new session. It also guarantees an identical connectivity when a case is read on a different number of compute nodes.

Note that writing and reading files in the hierarchical data format requires that the parallel version of ANSYS Fluent is used. For more details about HDF, see Reading and Writing Files Using Hierarchical Data Format (HDF) (p. 47).

When you write an overset case file in HDF and the case has not yet been initialized, ANSYS Fluent will by default establish the domain connectivity prior to writing the file. This can be disabled using the `define/overset-interfaces/options/update-before-case-write?` text command, which is available after you enable the expert tools using the `define/overset-interfaces/options/expert` command.

5.6. Checking the Mesh

The mesh checking capability in ANSYS Fluent examines various aspects of the mesh, including the mesh topology, periodic boundaries, simplex counters, and (for axisymmetric cases) node position with respect to the x axis, and provides a mesh check report with details about domain extents, statistics related to cell volume and face area, and information about any problems associated with the mesh. You can check the mesh by clicking the **Check** button in the **General** task page.

Setup → **General** → **Check**

Important

It is generally a good idea to check your mesh right after reading it into Fluent, in order to detect any mesh trouble before you get started with the problem setup.

The mesh check examines the topological information, beginning with the number of faces and nodes per cell. A triangular cell (2D) should have 3 faces and 3 nodes, a tetrahedral cell (3D) should have 4 faces and 4 nodes, a quadrilateral cell (2D) should have 4 faces and 4 nodes, and a hexahedral cell (3D) should have 6 faces and 8 nodes. Polyhedral cells (3D) will have an arbitrary number of faces and nodes.

Next, the face handedness and face node order for each zone is checked. The zones should contain all right-handed faces, and all faces should have the correct node order.

The last topological verification is checking the element-type consistency. If a mesh does not contain mixed elements (quadrilaterals and triangles or hexahedra and tetrahedra), ANSYS Fluent will determine that it does not need to keep track of the element types. By doing so, it can eliminate some unnecessary work.

For axisymmetric cases, the number of nodes below the x axis is listed. Nodes below the x axis are forbidden for axisymmetric cases, since the axisymmetric cell volumes are created by rotating the 2D cell volume about the x axis; therefore nodes below the x axis would create negative volumes.

For solution domains with rotationally periodic boundaries, the minimum, maximum, average, and prescribed periodic angles are computed. A common mistake is to specify the angle incorrectly. For domains with translationally periodic boundaries, the boundary information is checked to ensure that the boundaries are truly periodic.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Finally, the simplex counters are verified. The actual numbers of nodes, faces, and cells that Fluent has constructed are compared to the values specified in the corresponding header declarations in the mesh file. Any discrepancies are reported.

5.6.1. Mesh Check Report
5.6.2. Repairing Meshes

5.6.1. Mesh Check Report

When you click the **Check** button in the **General** task page, a mesh check report will be displayed in the console. The following is a sample of a successful output :

```
  Mesh Check

  Domain Extents:
    x-coordinate: min (m) = -4.000000e-002, max (m) = 2.550000e-001
    y-coordinate: min (m) = 0.000000e+000, max (m) = 2.500000e-002
  Volume statistics:
    minimum volume (m3): 2.463287e-009
    maximum volume (m3): 4.508038e-007
     total volume (m3): 4.190433e-004
    minimum 2d volume (m3): 3.000589e-007
    maximum 2d volume (m3): 3.019523e-006
  Face area statistics:
    minimum face area (m2): 4.199967e-004
    maximum face area (m2): 2.434403e-003
  Checking mesh.....................
 Done.
```

The mesh check report begins by listing the domain extents. The domain extents include the minimum and maximum x, y, and z coordinates in meters.

Then the volume statistics are provided, including the minimum, maximum, and total cell volume in m^3. A negative value for the minimum volume indicates that one or more cells have improper connectivity. Cells with a negative volume can often be identified using the **Iso-Value Adaption** dialog box to mark them for adaption and view them in the graphics window. For more information on creating and viewing isovalue adaption registers, see Isovalue Adaption (p. 1691). You *must* eliminate these negative volumes before continuing the flow solution process.

Next, the mesh report lists the face area statistics, including the minimum and maximum areas in m^2. A value of 0 for the minimum face area indicates that one or more cells have degenerated. As with negative volume cells, you *must* eliminate such faces. It is also recommended to correct cells that have nonzero face areas, if the values are very small.

Besides the information about the domain extents and statistics, the mesh check report will also display warnings based on the results of the checks previously described. You can specify that the mesh check report displays more detailed information about the various checks and the mesh failures, by using the following text command prior to performing the mesh check:

`mesh` → `check-verbosity`

You will then be prompted to enter the level of verbosity for the mesh check report. The possible levels include:

- `0`

  This is the default level, and only notifies you that checks are being performed (for example, **Checking mesh...**). The report will look like the previous example; while warnings will be displayed below the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

domain extents and statistics, the names of the individual checks will not be listed as they are conducted.

- 1

  This level lists the individual checks as they are performed (for example, **Checking right-handed cells**). Any warnings that result will be displayed immediately below the check that produced it.

- 2

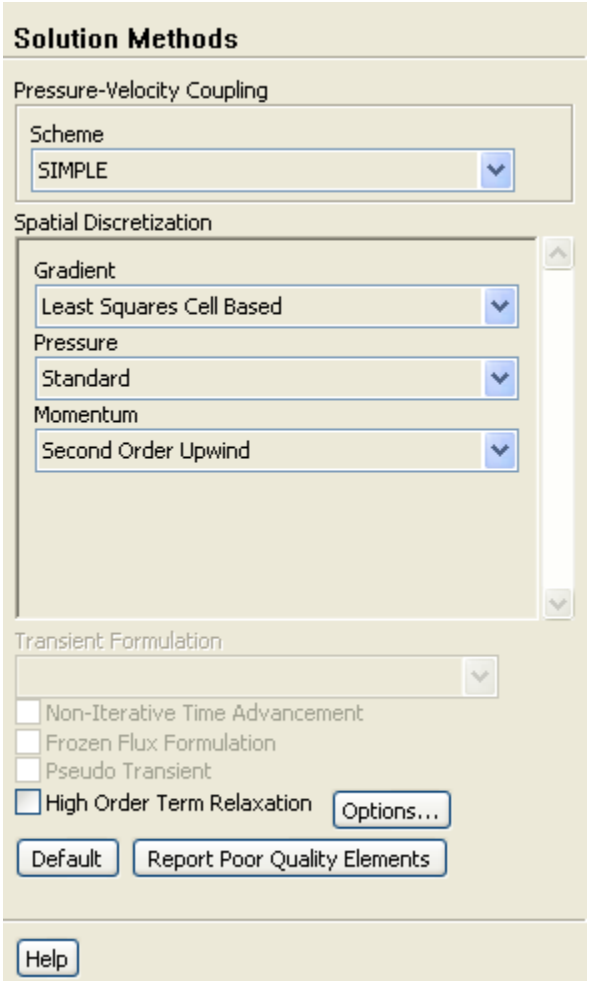
  This level provides the maximum information about the mesh check. The report will list the individual checks as they are performed (for example, **Checking right-handed cells**); any warnings that result will be displayed immediately below the check that produced it. Additional details about the check failure will also be displayed, such as the location of the problem or the affected cells.

5.6.2. Repairing Meshes

If the mesh check report indicates a mesh problem or if you receive warnings, you can investigate the extent of the problem by printing the poor element statistics in the console. This can be accomplished using the **Report Poor Quality Elements** button, located at the bottom of the **Solution Methods** task page (Figure 5.59: The Solution Methods Task Page (p. 195)). Note that this button is only available when the mesh has poor quality elements. For more information about the **Report Poor Quality Elements** feature, see Robustness on Meshes of Poor Quality (p. 1671).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.59: The Solution Methods Task Page



Alternatively, you can print the poor element statistics via the following text command:

`mesh → repair-improve → report-poor-elements`

You can also visualize the invalid and poor elements: first, mark them by using the **Mesh...** and **Mark Poor Elements** selections from the **Iso-Values of** drop-down lists of the **Iso-Value Adaption** dialog box (see Performing Isovalue Adaption (p. 1692) for further details). Then, display them in the graphics window using the **Manage Adaption Registers** dialog box, as described in Manipulating Adaption Registers (p. 1701). Similarly, you can display them using the **Contours** dialog box, by selecting **Mesh...** and **Mark Poor Elements** from the **Contours of** drop-down lists. In either case, a value of 1 is assigned to the cells that are identified as invalid or poor, as well as the cells that are adjacent to the face of an invalid or poor cell, and a value of 0 is assigned to all other cells.

The mesh check report will indicate if the mesh has problems that must be repaired, such as left-handed faces and/or faces that have the wrong node order. If repairable failures are detected by the mesh check, the **Repair** button in the **Setting Up Domain** ribbon tab (**Mesh** group box) becomes available:

Setting Up Domain → Mesh → Repair

Using the **Repair** button is the simplest way to attempt to correct your mesh problems. It will attempt to correct a number of problems identified by the mesh check, including cells that have:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- the wrong node order
- the wrong face handedness or that are not convex
- faces that are small or nonexistent
- very poor quality (see Mesh Quality (p. 131) for additional details)

Note that by default, the **Repair** button will only adjust the positions of interior nodes. If you want to also modify the nodes on the boundaries of the mesh, use the following text command *before* you repair the mesh:

`mesh → repair-improve → allow-repair-at-boundaries`

The **Repair** button may convert degenerate cells into polyhedra, based on skewness criteria (for more information on how cells are converted, see Converting Skewed Cells to Polyhedra (p. 204)). If you want to ensure that there are no polyhedra in the repaired mesh (for example, if you if plan on performing mesh adaption), you must disable such conversions using the following text command *before* you repair the mesh:

`mesh → repair-improve → include-local-polyhedra-conversion-in-repair`

If you would like to only attempt to improve the poor quality cells, you can use the following text command:

`mesh → repair-improve → improve-quality`

You can use the `improve-quality` text command multiple times, until the mesh is improved to your satisfaction. For greater control over the degree to which the mesh is improved, you can perform quality-based smoothing (as described in Smoothing (p. 230)).

It should be noted that both the **Repair** button and the `improve-quality` text command can be CPU intensive, if there are a large number of poor quality cells in the mesh. If you are not as concerned about cell quality, you can attempt to fix only the left-handed faces and faces with the wrong node order. Begin by repairing the node order with the following text command:

`mesh → repair-improve → repair-face-node-order`

Because the left-handed faces may be a result of improper face node order, the previous text command may resolve both issues at the same time. Be sure to perform another mesh check after entering the `repair-face-node-order` command, to see if the mesh has been fully repaired.

If at any point the mesh check reveals that the mesh contains left-handed faces without any node order issues, you can attempt to repair the face handedness by modifying the cell centroids with the following text command:

`mesh → repair-improve → repair-face-handedness`

Once again, perform a mesh check to see if the text command was successful. The `repair-face-handedness` text command is most effective for cells with high aspect ratios.

If the mesh check report includes a warning message such as

```
WARNING: node on face thread 2 has multiple shadows.
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

it indicates the existence of duplicate shadow nodes. This error occurs only in meshes with periodic-type walls. You can repair such a mesh using the following text command:

`mesh` → `repair-improve` → `repair-periodic`

If the interface is rotational periodic, you will be prompted for the rotation angle.

5.7. Reporting Mesh Statistics

There are several methods for reporting information about the mesh after it has been read into ANSYS Fluent. You can report the amount of memory used by the current problem, the mesh size, and statistics about the mesh partitions. Zone-by-zone counts of cells and faces can also be reported.

Information about mesh statistics is provided in the following sections:

5.7.1. Mesh Size
5.7.2. Memory Usage
5.7.3. Mesh Zone Information
5.7.4. Partition Statistics

5.7.1. Mesh Size

You can print out the numbers of nodes, faces, cells, and partitions in the mesh by clicking **Info** and selecting **Size**, in the **Setting Up Domain** ribbon tab (**Mesh** group box).

Setting Up Domain → **Mesh** → **Info** → **Size**

A partition is a piece of a mesh that has been segregated for parallel processing (see Parallel Processing (p. 2009)).

A sample of the resulting output follows:

```
Mesh Size

Level     Cells       Faces       Nodes        Partitions
    0      7917       12247       4468                 1

2 cell zones, 11 face zones.
```

If you are interested in how the cells and faces are divided among the different zones, click **Info** and select **Zones**, in the **Setting Up Domain** ribbon tab (**Mesh** group box), as described in Mesh Zone Information (p. 199).

If you are using the density-based coupled explicit solver, the mesh information will be printed for each grid level. The grid levels result from creating coarse grid levels for the FAS multigrid convergence acceleration (see Full-Approximation Storage (FAS) Multigrid in the Theory Guide). A sample of the resulting output is shown below:

```
Mesh Size

Level   Cells    Faces    Nodes     Partitions
 0       7917    12247     4468              1
 1       1347     3658        0              1
 2        392     1217        0              1
 3        133      475        0              1
 4         50      197        0              1
 5         17       78        0              1
2 cell zones, 11 face zones.
```

5.7.2. Memory Usage

During an ANSYS Fluent session you may want to check the amount of memory used and allocated in the present analysis. ANSYS Fluent has a feature that will report the following information: the numbers of nodes, faces, cells, edges, and object pointers (generic pointers for various mesh and graphics utilities) that are used and allocated; the amount of array memory (scratch memory used for surfaces) used and allocated; and the amount of memory used by the solver process.

You can obtain this information by clicking **Info** and selecting **Memory Usage**, in the **Setting Up Domain** ribbon tab (**Mesh** group box).

Setting Up Domain → **Mesh** → **Info** → **Memory Usage**

The memory information will be different for Linux and Windows systems.

5.7.2.1. Linux Systems

On Linux systems, note the following definitions related to process memory information:

- Process static memory is essentially the size of the code itself.
- Process dynamic memory is the allocated heap memory used to store the mesh and solution variables.
- Process total memory is the sum of static and dynamic memory.

5.7.2.2. Windows Systems

On Windows systems, note the following definitions related to process memory information:

- Process physical memory is the allocated heap memory currently resident in RAM.
- Process virtual memory is the allocated heap memory currently swapped to the Windows system page file.
- Process total memory is the sum of physical and virtual memory.

Note the following:

- The memory information does not include the static (code) memory.
- In the serial version of ANSYS Fluent, the heap memory value includes storage for the solver (mesh and solution variables), and Cortex (GUI and graphics memory), since Cortex and the solver are contained in the same process.
- In the parallel version, Cortex runs in its own process, so the heap memory value includes storage for the mesh and solution variables only.

On Windows systems, you can also get more information on the ANSYS Fluent process (or processes) by using the Task Manager (see your Windows documentation for details). For the serial version, the process image name will be something like `fl1400s.exe`. For the parallel version, examples of process image names are as follows: `cx1400.exe` (Cortex) and `fl1400.exe` (solver host).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.7.3. Mesh Zone Information

You can print information in the console about the nodes, faces, and cells in each zone the by clicking **Info** and selecting **Zones**, in the **Setting Up Domain** ribbon tab (**Mesh** group box).

Setting Up Domain → Mesh → Info → Zones

The mesh zone information includes the total number of nodes and, for each face and cell zone, the number of faces or cells, the cell (and, in 3D, face) type (triangular, quadrilateral, and so on), the boundary condition type, and the zone ID. Sample output is shown below:

```
Zone sizes on domain 1:
21280 hexahedral cells, zone 4.
  532 quadrilateral velocity-inlet faces, zone 1.
  532 quadrilateral pressure-outlet faces, zone 2.
  1040 quadrilateral symmetry faces, zone 3.
  1040 quadrilateral symmetry faces, zone 7.
 61708 quadrilateral interior faces, zone 5.
   1120 quadrilateral wall faces, zone 6.
23493 nodes.
```

5.7.4. Partition Statistics

You can print mesh partition statistics in the console by clicking **Info** and selecting **Partitions**, in the **Setting Up Domain** ribbon tab (**Mesh** group box).

Setting Up Domain → Mesh → Info → Partitions

The statistics include the numbers of cells, faces, interfaces, and neighbors of each partition. See Interpreting Partition Statistics (p. 2052) for further details, including sample output.

5.8. Converting the Mesh to a Polyhedral Mesh

Since the ANSYS Fluent solver is face based, it supports polyhedral cells. The advantages that polyhedral meshes have shown over some of the tetrahedral or hybrid meshes is the lower overall cell count, almost 3-5 times lower for unstructured meshes than the original cell count. Currently, there are three options in ANSYS Fluent that allow you to convert your non-polyhedral cells to a polyhedra:

- Converting the entire domain into polyhedral cells (applicable only for meshes that contain tetrahedral and/or wedge cells).
- Converting skewed tetrahedral cells to polyhedral cells.
- Converting cells with hanging nodes/edges to polyhedral cells.

Information about polyhedral mesh conversion is provided in the following sections:

5.8.1. Converting the Domain to a Polyhedra
5.8.2. Converting Skewed Cells to Polyhedra
5.8.3. Converting Cells with Hanging Nodes / Edges to Polyhedra

5.8.1. Converting the Domain to a Polyhedra

Conversion of a mesh to polyhedra only applies to 3D meshes that contain tetrahedral and/or wedge cells.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

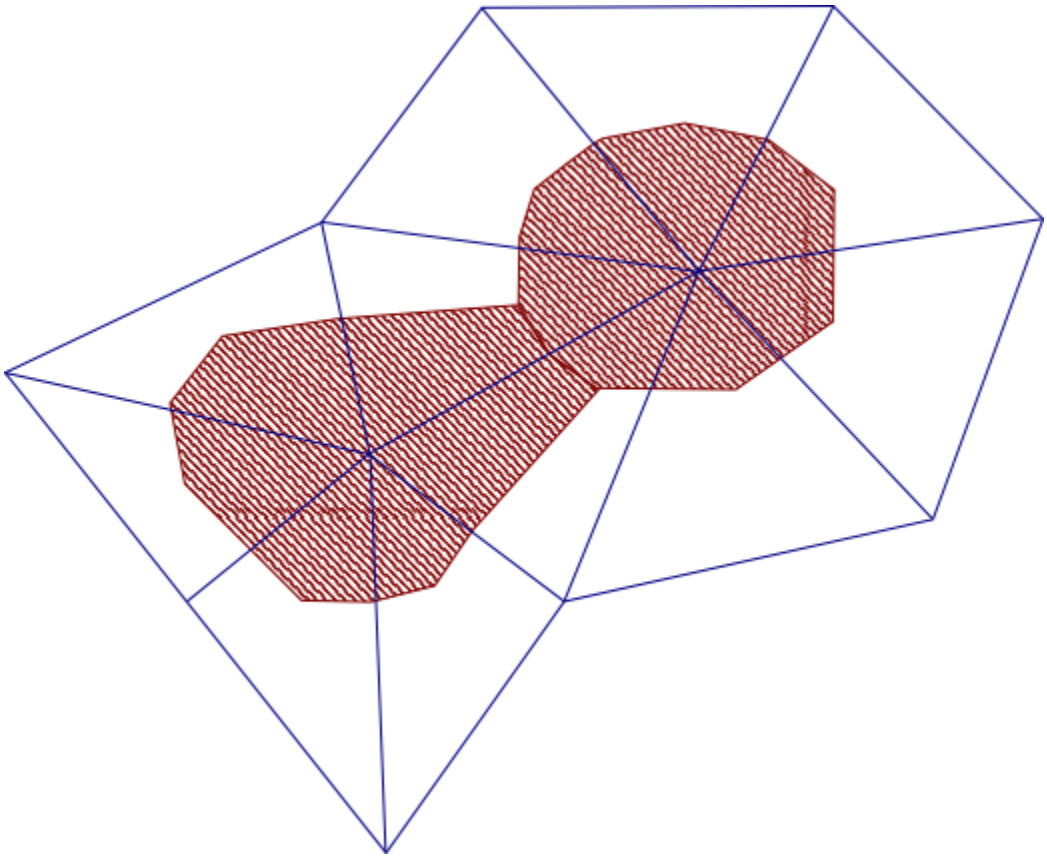
To begin the conversion process, ANSYS Fluent automatically decomposes each non-hexahedral cell into multiple sub-volumes called "duals" (the shaded regions seen in the 2D example in Figure 5.60: Connection of Edge Centroids with Face Centroids (p. 200)). Each dual is associated with one of the original nodes of the cell. These duals are then agglomerated into polyhedral cells around the original nodes. Therefore, the collection of duals from all cells sharing a particular node makes up each polyhedral cell (see Figure 5.61: A Polyhedral Cell (p. 201)). The node that is now within the polyhedral cell is no longer needed and is removed.

To better understand how duals are formed, you can consider the straightforward case of a tetrahedral mesh. Each of the cells are decomposed in the following manner: first, new edges are created on each face between the face centroid and the centroids of the edges of that face. Then, new faces are created within the cell by connecting the cell centroid to the new edges on each face. These interior faces establish the boundaries between the duals of a cell, and divide the cell into 4 sub-volumes. These dividing faces may be adjusted and merged with neighboring faces during the agglomeration process, in order to minimize the number of faces on the resultant polyhedral cell.

Note

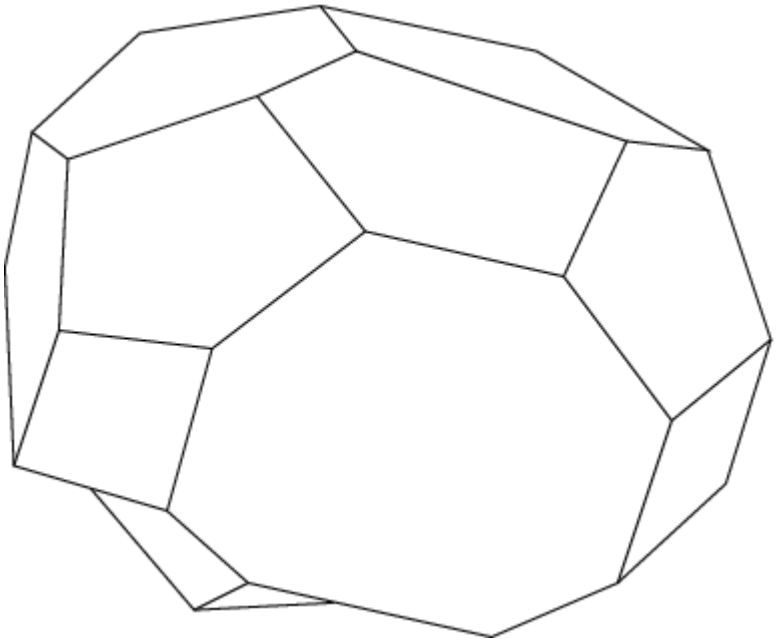
By default, ANSYS Fluent checks the aspect ratio of boundary layer cells and if the aspect ratios are high, ANSYS Fluent asks whether you want to preserve cells at the boundary layer (which could be useful for keeping cell counts low when retaining boundary features). You can override this default with the text command: `mesh/polyhedra/options/preserve-boundary-layer?` (for additional information about these options, see `mesh/` in the *Fluent Text Command List*).

Figure 5.60: Connection of Edge Centroids with Face Centroids



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

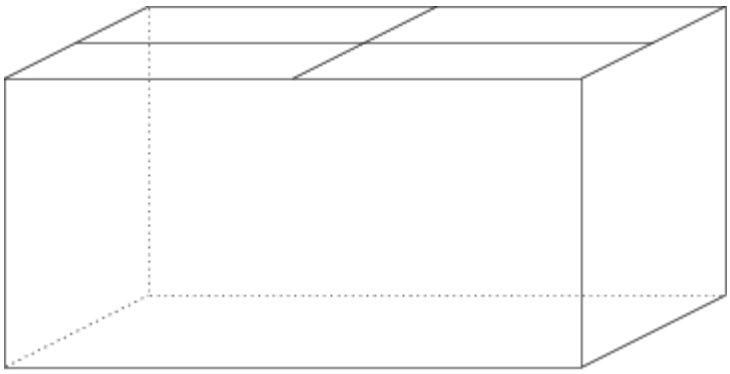
Figure 5.61: A Polyhedral Cell



Important

Hexahedral cells are not converted to polyhedra when the domain is converted, except when they border non-hexahedral cells. When the neighboring cell is reconfigured as polyhedra, the shared face of the hexahedral cell is decomposed into multiple faces as well, resulting in a polyhedral cell. In such a case the shape of the original hexahedral cell is preserved (that is the overall dimensions of the cell stay the same), but the converted cell has more than the original 6 faces (see Figure 5.62: A Converted Polyhedral Cell with Preserved Hexahedral Cell Shape (p. 201)).

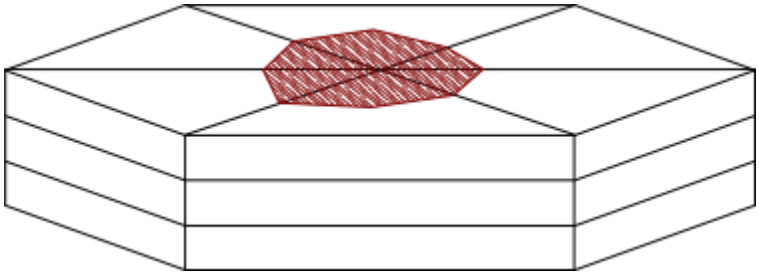
Figure 5.62: A Converted Polyhedral Cell with Preserved Hexahedral Cell Shape



Conversion proceeds in a slightly different manner in boundary layers that are modeled using thin wedge cells. These cells are decomposed in the plane of the boundary surface, but not in the direction normal to the surface. The resulting polyhedra will therefore preserve the thickness of the original wedge cells (Figure 5.63: Treatment of Wedge Boundary Layers (p. 202)). In most cases, the cell count in the new polyhedral boundary layer will be lower than the original boundary layer.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.63: Treatment of Wedge Boundary Layers



To convert the entire domain of your mesh, click **Make Polyhedra** in the **Setting Up Domain** ribbon tab (**Mesh** group box).

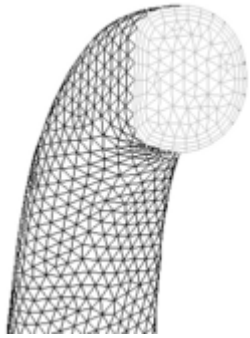
Setting Up Domain → Mesh → Make Polyhedra

The following is an example of the resulting message printed in the console:

```
  Setup conversion to polyhedra.
  Converting domain to polyhedra...

       Creating polyhedra zones.
       Processing face zones..........
       Processing cell zones...
       Building polyhedra mesh.................
       Optimizing polyhedra mesh.....
 >> Reordering domain using Reverse Cuthill-McKee method:
     zones, cells, faces, done.
  Bandwidth reduction = 1796/247 = 7.27
  Done.
```

Figure 5.64: The Original Tetrahedral Mesh (p. 202), the original tetrahedral mesh of a section of a manifold, is compared to Figure 5.65: The Converted Polyhedral Mesh (p. 203), which is the resulting mesh after the entire domain is converted to a polyhedra.

Figure 5.64: The Original Tetrahedral Mesh



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.65: The Converted Polyhedral Mesh

Note that by default, the surfaces (that is, manifold zones of type **interior**) will be lost during the conversion to polyhedra. If you would like to preserve any of these zones (in order to utilize them for postprocessing, for example), use the following text command prior to the conversion:

`mesh` → `polyhedra` → `options` → `preserve-interior-zones`

You will be prompted to enter a string of characters, and only those interior surfaces with a name that includes the string you specify will be preserved.

In parallel, mesh migration and reordering is automatically performed after the conversion is performed. If this process takes a long time and the solver will not be run immediately in the same Fluent session, you can disable it using the following text command:

`mesh` → `polyhedra` → `options` → `migrate-and-reorder?`

If migration/reordering is disabled, the new mesh partitioning will be applied when the mesh is read into a new Fluent session and it is recommended that you manually reorder at that time.

5.8.1.1. Limitations

Some limitations you will find with polyhedral meshes that you generally do not experience with other cell types include:

- Meshes that already contain polyhedral cells cannot be converted.
- Meshes with hanging nodes/edges will not be converted. This includes meshes that have undergone hanging node adaption (see Hanging Node Adaption in the Theory Guide), as well as CutCell meshes (generated by Workbench or the meshing mode of Fluent, or resulting from the CutCell zone remeshing method in the solution mode of Fluent) and hexcore meshes (generated when using the **Hex Core** meshing scheme in GAMBIT or the **Hexcore** menu option in the meshing mode of Fluent).
- The following mesh manipulation tools are not available for polyhedral meshes:
  - the `mesh/modify-zones/extrude-face-zone-delta` text command
  - the `mesh/modify-zones/extrude-face-zone-para` text command
  - skewness smoothing
  - swapping

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Meshes in which the domain has been converted to polyhedral cells are not eligible for adaption. For more information about adaption, see Adapting the Mesh (p. 1681).
- A mesh that is entirely comprised of polyhedral cells has limitations when it is to be used in a dynamic mesh problem: it cannot undergo dynamic layering, and the only remeshing method available is CutCell zone remeshing. Smoothing is allowed for polyhedral cells, and diffusion-based smoothing is recommended over spring-based smoothing (see Diffusion-Based Smoothing (p. 650) for details). The linearly elastic solid smoothing method is not compatible with polyhedral cells.
- Since polyhedral meshes can have a much higher ratio of nodes to cells then tet or hex meshes, the use of the node-based gradient method may result in a significant increase in memory consumption compared with other gradient methods.

5.8.2. Converting Skewed Cells to Polyhedra

Another method of cell agglomeration is the skewness-based cluster approach. This type of conversion is designed to convert only part of the domain. The objective is to convert only skewed tetrahedral cells above a specified cell equivolume skewness threshold into polyhedra. By converting the highly skewed tetrahedral cells, the quality of the mesh can be improved significantly.

A different algorithm is used for local conversion. This algorithm evaluates each highly skewed tetrahedral cell and all of the surrounding cells, to select an edge on the highly skewed cell that best matches criteria for cell agglomeration. Then all of the cells that share this edge are combined into a polyhedral cell. During the process, the data is interpolated from the original cells to the resultant polyhedra.

5.8.2.1. Limitations

There are certain limitations with this type of conversion:

- The following mesh manipulation tools are not available on polyhedral meshes:
  - the `mesh/modify-zones/extrude-face-zone-delta` text command
  - the `mesh/modify-zones/extrude-face-zone-para` text command
  - skewness smoothing
  - swapping will not affect polyhedral cells
- The polyhedral cells that result from the conversion are not eligible for adaption. For more information about adaption, see Adapting the Mesh (p. 1681).
- Only tetrahedral cells are converted, as all other cells are skipped.
- Meshes with hanging nodes / edges will not be converted. This includes meshes that have undergone hanging node adaption (see Hanging Node Adaption in the Theory Guide), as well as CutCell meshes (generated by Workbench or the meshing mode of Fluent, or resulting from the CutCell zone remeshing method in the solution mode of Fluent) and hexcore meshes (generated when using the **Hex Core** meshing scheme in GAMBIT or the **Hexcore** menu option in the meshing mode of Fluent). Note that if the mesh is a CutCell / hexcore mesh in which the transitional cells have been converted to polyhedra, then it does not have hanging nodes / edges and can therefore be converted.
- The polyhedral cells that result from this conversion have the following limitations with regard to the update methods available for dynamic mesh problems:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - When applying dynamic layering to a cell zone, you cannot have polyhedral cells adjacent to the moving face zone.

  - None of the remeshing methods except for CutCell zone remeshing will modify polyhedral cells.

  Note that smoothing is allowed for polyhedral cells, and diffusion-based smoothing is recommended over spring-based smoothing (see Diffusion-Based Smoothing (p. 650) for details). The linearly elastic solid smoothing method is not compatible with polyhedral cells.

5.8.3. Converting Cells with Hanging Nodes / Edges to Polyhedra

ANSYS Fluent provides a text command that allows you to convert cells that have hanging nodes / edges into polyhedra. Such a conversion may be done in order to prevent errors associated with interior walls (see Using the `tpoly` Filter to Remove Hanging Nodes/Edges (p. 143)). Each of the converted polyhedra preserve the shape of the original cell (that is, the overall dimensions of each cell stay the same), but the number of faces associated with each cell increases (see Figure 5.62: A Converted Polyhedral Cell with Preserved Hexahedral Cell Shape (p. 201) for an example). Such a conversion may be helpful for CutCell meshes (generated by Workbench or the meshing mode of Fluent, or resulting from the CutCell zone remeshing method in the solution mode of Fluent), as well as hexcore meshes (generated when using the **Hex Core** meshing scheme in GAMBIT or the **Hexcore** menu option in the meshing mode of Fluent). You could also convert the cells with hanging nodes in a mesh that has undergone adaption (see Adapting the Mesh (p. 1681)), but you would need to be sure that no further refinement/coarsening of the mesh will be necessary.

To convert the cells with hanging nodes / edges, use the following text command:

`mesh` → `polyhedra` → `convert-hanging-nodes`

5.8.3.1. Limitations

There are certain limitations with this type of conversion:

- The following mesh manipulation tools are not available on polyhedral meshes:

  - the `mesh/modify-zones/extrude-face-zone-delta` text command

  - the `mesh/modify-zones/extrude-face-zone-para` text command

  - skewness smoothing

  - swapping will not affect polyhedral cells

- The polyhedral cells that result from the conversion are not eligible for adaption.

- The polyhedral cells that result from this conversion have the following limitations with regard to the update methods available for dynamic mesh problems:

  - When applying dynamic layering to a cell zone, you cannot have polyhedral cells adjacent to the moving face zone.

  - None of the remeshing methods except for CutCell zone remeshing will modify polyhedral cells.

  Note that smoothing is allowed for polyhedral cells, and diffusion-based smoothing is recommended over spring-based smoothing (see Diffusion-Based Smoothing (p. 650) for details). The linearly elastic solid smoothing method is not compatible with polyhedral cells.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.9. Modifying the Mesh

There are several ways in which you can modify or manipulate the mesh after it has been read into ANSYS Fluent. You can scale or translate the mesh, copy, merge, or separate zones, create or slit periodic zones, fuse boundaries, and smooth and swap faces. Methods for partitioning meshes to be used in a parallel solver are discussed in Mesh Partitioning and Load Balancing (p. 2030).

Important

Whenever you modify the mesh, you should be sure to save a new case file (and a data file, if data exists). If you have old data files that you would like to be able to read in again, be sure to retain the original case file as well, as the data in the old data files may not correspond to the new case file.

Information about mesh manipulation is provided in the following sections:

5.9.1. Merging Zones
5.9.2. Separating Zones
5.9.3. Fusing Face Zones
5.9.4. Creating Conformal Periodic Zones
5.9.5. Slitting Periodic Zones
5.9.6. Slitting Face Zones
5.9.7. Orienting Face Zones
5.9.8. Extruding Face Zones
5.9.9. Replacing, Deleting, Deactivating, and Activating Zones
5.9.10. Copying Cell Zones
5.9.11. Replacing the Mesh
5.9.12. Managing Adjacent Zones
5.9.13. Reordering the Domain
5.9.14. Scaling the Mesh
5.9.15. Translating the Mesh
5.9.16. Rotating the Mesh
5.9.17. Improving the Mesh by Smoothing and Swapping

5.9.1. Merging Zones

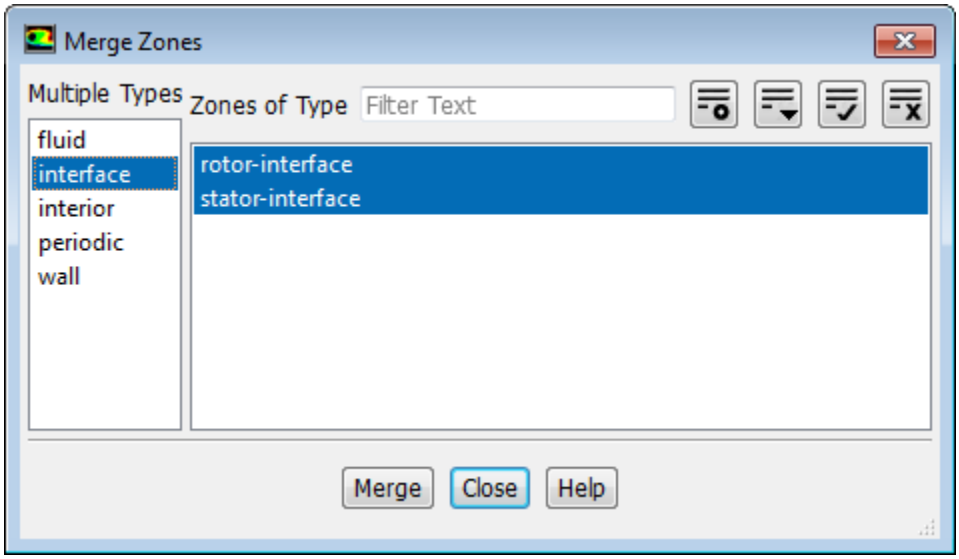
To simplify the solution process, you may want to merge zones. Merging zones involves combining multiple zones of similar type into a single zone. Setting boundary conditions and postprocessing may be easier after you have merged similar zones.

Zone merging is performed in the Merge Zones Dialog Box (p. 2730) (Figure 5.66: The Merge Zones Dialog Box (p. 207)).

Setting Up Domain → Zones → Combine → Merge...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.66: The Merge Zones Dialog Box



5.9.1.1. When to Merge Zones

ANSYS Fluent allows you to merge zones of similar type into a single zone. This is not necessary unless the number of zones becomes prohibitive to efficient setup or postprocessing of the numerical analysis. For example, setting the same boundary condition parameters for a large number of zones can be time-consuming and may introduce inconsistencies. In addition, the postprocessing of the data often involves surfaces generated using the zones. A large number of zones often translates into a large number of surfaces that must be selected for the various display options, such as color contouring. Fortunately, surfaces can also be merged (see Grouping, Renaming, and Deleting Surfaces (p. 1734)), minimizing the negative impact of a large number of zones on postprocessing efficiency.

Although merging zones can be helpful, there may be cases where you will want to retain a larger number of zones. Since the merging process is not fully reversible, a larger number of zones provides more flexibility in imposing boundary conditions. Although a large number of zones can make selection of surfaces for display tedious, it can also provide more choices for rendering the mesh and the flow-field solution. For instance, it can be difficult to render an internal flow-field solution. If the outer domain is composed of several zones, the meshes of subsets of these zones can be plotted along with the solution to provide the relationship between the geometry and solution field. Merging zones may also adversely affect dynamic zones and mesh interfaces.

5.9.1.2. Using the Merge Zones Dialog Box

The procedure for merging multiple zones of the same type into a single zone is as follows:

1. Select the zone type in the **Multiple Types** list. This list contains all the zone types for which there are multiple zones. When you choose a type from this list, the corresponding zones will appear in the **Zones of Type** list.

2. Select two or more zones in the **Zones of Type** list.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click the **Merge** button to merge the selected zones. Note that if your case file has dynamic zones or mesh interfaces, the **Warning** dialog box will open before the merge is initiated, allowing you to specify whether you want to delete such zones or interfaces first (see Warning Dialog Box (p. 2841) for details).

Important

Remember to save a new case file (and a data file, if data exists).

5.9.2. Separating Zones

Upon reading a mesh file, ANSYS Fluent automatically performs zone separations in two conditions. If a face zone is attached to multiple cells zones in the preprocessor, the face zone will be separated so that each one is attached to only one cell zone. Furthermore, if you have defined an internal face as a wall type, an additional shadow wall zone will be generated (for example, for a wall named **baffle**, a shadow wall zone named **baffle-shadow** will be generated).

There are several methods available in ANSYS Fluent that allow you to manually separate a single face or cell zone into multiple zones of the same type. If your mesh contains a zone that you want to break up into smaller portions, you can make use of these features. For example, if you created a single wall zone when generating the mesh for a duct, but you want to specify different temperatures on specific portions of the wall, you will need to break that wall zone into two or more wall zones. If you plan to solve a problem using the sliding mesh model or multiple reference frames, but you forgot to create different fluid zones for the regions moving at different speeds, you will need to separate the fluid zone into two or more fluid zones.

Important

- After performing any of these separations, you should save a new case file. If data exists, it is automatically assigned to the proper zones when separation occurs, so you should also write a new data file. The old data cannot be read on top of the case file in which the zones have changed.
- The maximum number of zones into which you can separate any one face zone or cell zone is 32.

There are four ways to separate face zones and two ways to separate cell zones. The face separation methods will be described first, followed by the cell separation tools. Slitting (decoupling) of periodic zones is discussed in Slitting Periodic Zones (p. 215).

Note that all of the separation methods allow you to report the result of the separation before you commit to performing it.

5.9.2.1. Separating Face Zones

For more information, see the following sections:
5.9.2.1.1. Methods for Separating Face Zones
5.9.2.1.2. Inputs for Separating Face Zones

5.9.2.1.1. Methods for Separating Face Zones

For geometries with sharp corners, it is often easy to separate face zones based on significant angle. Faces with normal vectors that differ by an angle greater than or equal to the specified significant angle

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

will be placed in different zones. For example, if your mesh consists of a cube, and all 6 sides of the cube are in a single wall zone, you would specify a significant angle of 89°. Since the normal vector for each cube side differs by 90° from the normals of its adjacent sides, each of the 6 sides will be placed in a different wall zone.

If you have a small face zone and would like to put each face in the zone into its own zone, you can do so by separating the faces based on face. Each individual face (triangle, quad, or polygon) will be separated into different zones.

You can also separate face zones based on the marks stored in adaption registers. For example, you can mark cells for adaption based on their location in the domain (region adaption), their boundary closeness (boundary adaption), isovalues of some variable, or any of the other adaption methods discussed in Adapting the Mesh (p. 1681). When you specify which register is to be used for the separation of the face zone, all faces of cells that are marked will be placed into a new face zone. (Use the Manage Adaption Registers Dialog Box (p. 2728) to determine the ID of the register you want to use.)

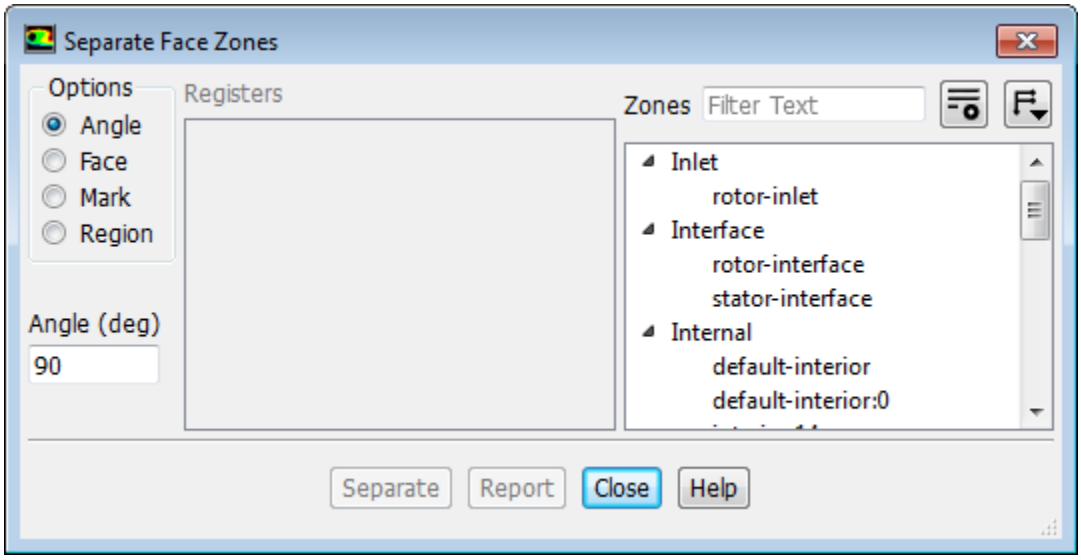
Finally, you can separate face zones based on contiguous regions. For example, when you use coupled wall boundary conditions you need the faces on the zone to have a consistent orientation. Consistent orientation can only be guaranteed on contiguous regions, so you may need to separate face zones to allow proper boundary condition specification.

5.9.2.1.2. Inputs for Separating Face Zones

To break up a face zone based on angle, face, adaption mark, or region, use the Separate Face Zones Dialog Box (p. 2794) (Figure 5.67: The Separate Face Zones Dialog Box (p. 209)).

Setting Up Domain → Zones → Separate → Faces...

Figure 5.67: The Separate Face Zones Dialog Box



Important

If you are planning to separate face zones, you should do so before performing any adaptions using the (default) hanging node adaption method. Face zones that contain hanging nodes cannot be separated.

The steps for separating faces are as follows:

1. Select the separation method (**Angle**, **Face**, **Mark**, or **Region**) under **Options**.
2. Specify the face zone to be separated in the **Zones** list.
3. If you are separating by face or region, skip to the next step. Otherwise, do one of the following:
   - If you are separating faces by angle, specify the significant angle in the **Angle** field.
   - If you are separating faces by mark, select the adaption register to be used in the **Registers** list.
4. (optional) To check what the result of the separation will be before you actually separate the face zone, click the **Report** button. The report will look like the following example:

```
45 faces in contiguous region 0
30 faces in contiguous region 1
11 faces in contiguous region 2
14 faces in contiguous region 3
Separates zone 4 into 4 zone(s).
```

5. To separate the face zone, click the **Separate** button. A report will be printed in the console like the following example:

```
45 faces in contiguous region 0
30 faces in contiguous region 1
11 faces in contiguous region 2
14 faces in contiguous region 3
Separates zone 4 into 4 zone(s).
Updating new zone information ...
   created new zone wall-4:001 from wall-4
   created new zone wall-4:002 from wall-4
   created new zone wall-4:010 from wall-4
done.
```

Important

When you separate the face zone by adaption mark, you may sometimes find that a face of a corner cell will be placed in the wrong face zone. You can usually correct this problem by performing an additional separation, based on angle, to move the offending face to a new zone. You can then merge this new zone with the zone in which you want the face to be placed, as described in Merging Zones (p. 206).

5.9.2.2. Separating Cell Zones

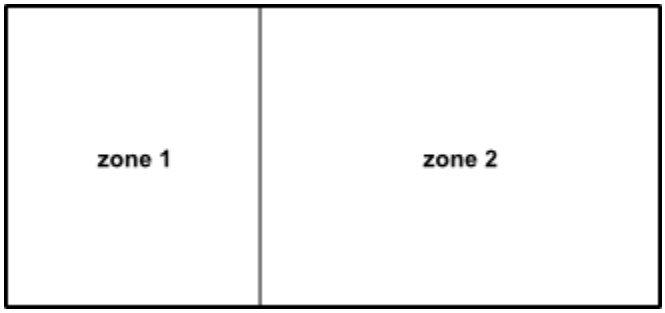
For more information, see the following sections:
5.9.2.2.1. Methods for Separating Cell Zones
5.9.2.2.2. Inputs for Separating Cell Zones

5.9.2.2.1. Methods for Separating Cell Zones

If you have two or more enclosed cell regions sharing internal boundaries (as shown in Figure 5.68: Cell Zone Separation Based on Region (p. 211)), but all of the cells are contained in a single cell zone, you can separate the cells into distinct zones using the separation-by-region method. Note that if the shared internal boundary is of type **interior**, you must change it to another double-sided face zone type (**fan**, **radiator**, and so on) prior to performing the separation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.68: Cell Zone Separation Based on Region



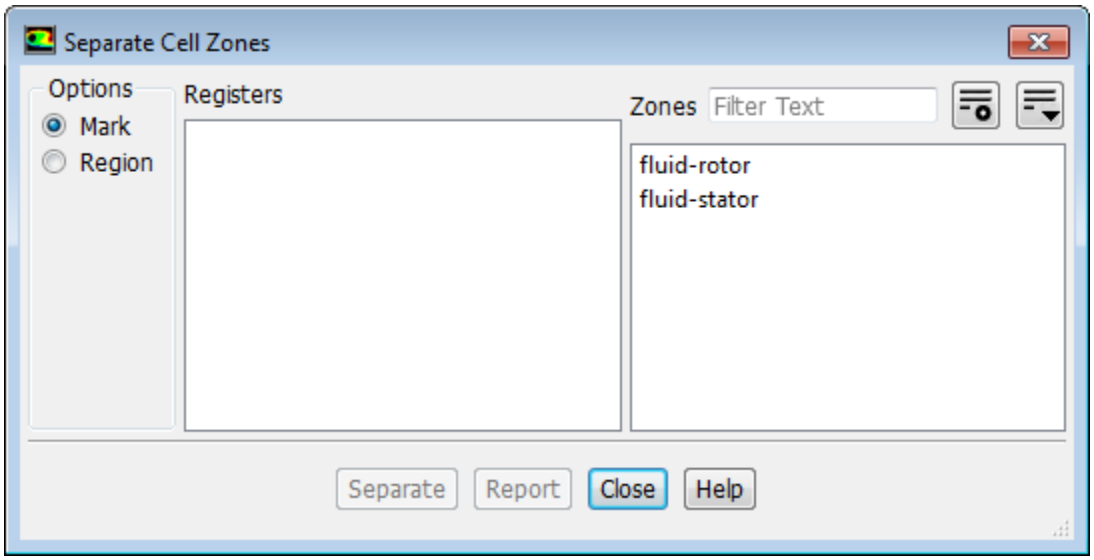
You can also separate cell zones based on the marks stored in adaption registers. You can mark cells for adaption using any of the adaption methods discussed in Adapting the Mesh (p. 1681) (for example, you can mark cells with a certain isovalue range or cells inside or outside a specified region). When you specify which register is to be used for the separation of the cell zone, cells that are marked will be placed into a new cell zone. (Use the Manage Adaption Registers Dialog Box (p. 2728) to determine the ID of the register you want to use.)

5.9.2.2.2. Inputs for Separating Cell Zones

To break up a cell zone based on region or adaption mark, use the Separate Cell Zones Dialog Box (p. 2793) (Figure 5.69: The Separate Cell Zones Dialog Box (p. 211)).

Setting Up Domain → Zones → Separate → Cells...

Figure 5.69: The Separate Cell Zones Dialog Box



Important

If you are planning to separate cell zones, you should do so before performing any adaptions using the (default) hanging node adaption method. Cell zones that contain hanging nodes cannot be separated.

The steps for separating cells are as follows:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the separation method (**Mark** or **Region**) under **Options**.

2. Specify the cell zone to be separated in the **Zones** list.

3. If you are separating cells by mark, select the adaption register to be used in the **Registers** list.

4. (optional) To check what the result of the separation will be before you actually separate the cell zone, click the **Report** button. The report will look like this:

```
Separates zone 14 into two zones, with 1275 and 32 cells.
```

5. To separate the cell zone, click the **Separate** button. ANSYS Fluent will print the following information:

```
Separates zone 14 into two zones, with 1275 and 32 cells.
No faces marked on thread, 2
No faces marked on thread, 3
No faces marked on thread, 1
No faces marked on thread, 5
No faces marked on thread, 7
No faces marked on thread, 8
No faces marked on thread, 9
No faces marked on thread, 61
Separates zone 62 into two zones, with 1763 and 58 faces.
All faces marked on thread, 4
No faces marked on thread, 66
Moved 32 cells from cell zone 14 to zone 10
Updating new zone information ...
   created new zone interior-4:010 from interior-4
   created new zone interior-6:009 from interior-6
   created new zone fluid-14:008 from fluid-14
done.
```

As shown in the example above, separation of a cell zone will often result in the separation of face zones as well. When you separate by mark, faces of cells that are moved to a new zone will be placed in a new face zone. When you separate by region, faces of cells that are moved to a new zone will not necessarily be placed in a new face zone.

If you find that any faces are placed incorrectly, see the comment above, at the end of the inputs for face zone separation.

5.9.3. Fusing Face Zones

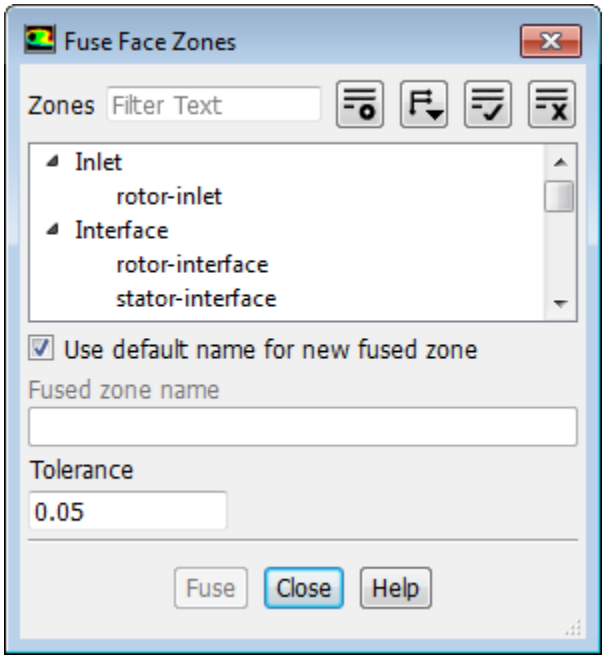
The face-fusing utility is a convenient feature that can be used to fuse boundaries (and merge duplicate nodes and faces) created by assembling multiple mesh regions. When the domain is divided into subdomains and the mesh is generated separately for each subdomain, you will combine the subdomains into a single file before reading the mesh into Fluent. For details, see Reading Multiple Mesh/Case/Data Files (p. 145). This situation could arise if you generate each block of a multiblock mesh separately and save it to a separate mesh file. Another possible scenario is that you decided, during mesh generation, to save the mesh for each part of a complicated geometry as a separate part file. (Note that the mesh node locations need not be identical at the boundaries where two subdomains meet; see Non-Conformal Meshes (p. 150) for details.)

The **Fuse Face Zones** dialog box (Figure 5.70: The Fuse Face Zones Dialog Box (p. 213)) allows you to merge the duplicate nodes and delete these artificial internal boundaries.

Setting Up Domain → Zones → Combine → Fuse...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.70: The Fuse Face Zones Dialog Box



The boundaries on which the duplicate nodes lie are assigned zone ID numbers (just like any other boundary) when the mesh files are combined, as described in Reading Multiple Mesh/Case/Data Files (p. 145). You need to keep track of the zone ID numbers when `tmerge` or the meshing mode of Fluent reports its progress or, after the complete mesh is read in, display all boundary mesh zones and use the mouse-probe button to determine the zone names (see Controlling the Mouse Button Functions (p. 1800) for information about the mouse button functions). Alternatively, you can specify a name for the fused zone.

5.9.3.1. Inputs for Fusing Face Zones

The steps for fusing face zones are as follows:

1. Select the zones to be fused in the **Zones** list.

   Important

   When using the Fuse Face Zone utility, each of the selected face zones must be fused in its entirety with another face zone having the same connectivity. Partial fusing of face zones is not supported.

2. (Optional) If you would like to specify a name for the fused zone rather than use an automatically generated name (for instance, to preserve the original name of one of the face zones), disable **Use default name for new fused zone** and enter the desired name.

3. Click the **Fuse** button to fuse the selected zones.

If all of the appropriate faces do not get fused using the default **Tolerance**, you should increase it and attempt to fuse the zones again. (This tolerance is the same as the matching tolerance discussed in Creating Conformal Periodic Zones (p. 214).) The **Tolerance** should not exceed 0.5, or you may fuse the wrong nodes.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also fuse zones using the mesh/modify-zones/fuse-face-zones text command, as shown in the following example.

```
/mesh/modify-zones > fuse-face-zones

()
Zone to fuse(1) [()]  w1.top

Zone to fuse(2) [()]  w3

Zone to fuse(3) [()]  [Enter]

Choose from list (automatic w3 w1.top) or pick your own
Enter the fused zone name: [automatic]  w3

  Name of zone 14 is changed into w3.

Fused list of zones successfully.
```

Important

Remember to save a new case file (and a data file, if data exists) after fusing faces.

5.9.3.1.1. Fusing Zones on Branch Cuts

Meshes imported from structured mesh generators or solvers (such as FLUENT 4) can often be O-type or C-type meshes with a reentrant branch cut where coincident duplicate nodes lie on a periodic boundary. Since ANSYS Fluent uses an unstructured mesh representation, there is no reason to retain this artificial internal boundary. (You can preserve these periodic boundaries and the solution algorithm will solve the problem with periodic boundary conditions.)

To fuse this periodic zone with itself, you must first slit the periodic zone, as described in Slitting Periodic Zones (p. 215). This will create two symmetry zones that you can fuse using the procedure above.

Note that if you need to fuse portions of a non-periodic zone with itself, you must use the mesh/modify-zones/fuse-face-zones text command.

mesh → modify-zones → fuse-face-zones

This command will prompt you for the name or ID of each zone to be fused. (You will enter the same zone twice.) To change the node tolerance, use the mesh/modify-zones/matching-tolerance text command.

5.9.4. Creating Conformal Periodic Zones

ANSYS Fluent allows you to set up periodic boundaries using either conformal or non-conformal periodic zones. You can assign periodicity to your mesh by coupling a pair of face zones. If the two zones have identical node and face distributions, you can create a conformal periodic zone. If the two zones are not identical at the boundaries within a specified tolerance, then you can create a periodic zone at a non-conformal mesh interface (Non-Conformal Meshes (p. 150)).

Important

Remember to save a new case file (and a data file, if data exists) after creating or slitting a periodic boundary.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To create conformal periodic boundaries, you will use the `mesh/modify-zones/make-periodic` text command.

`mesh` → `modify-zones` → `make-periodic`

You will need to specify the two face zones that will comprise the periodic pair (you can enter their full names or just their IDs), and indicate whether they are rotationally or translationally periodic. The order in which you specify the periodic zone and its matching shadow zone is not significant.

```
  /mesh/modify-zones>  make-periodic

Periodic zone [()] 1
Shadow zone [()] 4
Rotational periodic? (if no, translational) [yes] n
Create periodic zones? [yes] yes
Auto detect translation vector? [yes] yes

    computed translation deltas: -2.000000 -2.000000
    all 10 faces matched for zones 1 and 4.

    zone 4 deleted

    created periodic zones.
```

When you create a conformal periodic boundary, Fluent will check to see if the faces on the selected zones "match" (that is, whether or not the nodes on corresponding faces are coincident). The matching tolerance for a face is a fraction of the minimum edge length of the face. If the periodic boundary creation fails, you can change the matching tolerance using the `mesh/modify-zones/matching-tolerance` text command, but it should not exceed 0.5 or you may match up the periodic zones incorrectly and corrupt the mesh.

`mesh` → `modify-zones` → `matching-tolerance`

For information about creating non-conformal periodic boundaries, see Using a Non-Conformal Mesh in ANSYS Fluent (p. 164).

5.9.5. Slitting Periodic Zones

If you want to decouple the zones in a periodic pair, you can use the `mesh/modify-zones/slit-periodic` text command.

`mesh` → `modify-zones` → `slit-periodic`

You will specify the periodic zone's name or ID, and Fluent will decouple the two zones in the pair (the periodic zone and its shadow) and change them to two symmetry zones:

```
  /mesh/modify-zones> slit-periodic

Periodic zone [()] periodic-1
Slit periodic zone? [yes] yes

   Slit periodic zone.
```

5.9.6. Slitting Face Zones

The face-zone slitting feature has two uses:

- You can slit a single boundary zone of any double-sided type (that is, any boundary zone that has cells on both sides of it) into two distinct zones.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You can slit a coupled wall zone into two distinct, uncoupled wall zones.

When you slit a face zone, Fluent will duplicate all faces and nodes, except those nodes that are located at the ends (2D) or edges (3D) of the zone. One set of nodes and faces will belong to one of the resulting boundary zones and the other set will belong to the other zone. The only ill effect of the shared nodes at each end is that you may see some inaccuracies at those points when you graphically display solution data with a slit boundary. (Note that if you adapt the boundary after slitting, you will not be able to fuse the boundaries back together again.)

Before you can slit, you first need to select the zone in the **Boundary Condition** task page and change the **Type** to **wall**. Upon changing the **Type** to **wall**, another shadow zone will be created (that is, if the original zone is called **rad-outlet**, another zone called **rad-outlet-shadow** will be created). Then you can apply the `mesh/modify-zones/slit-face-zone` text command on either of the walls (that is, **rad-outlet** or **rad-outlet-shadow**) to separate them into two distinct walls.

Important

When a wall and wall-shadow pair is created by slitting a face zone, the wall will inherit the zone id number and face orientation of the original zone. A new zone id number will be created and assigned to the wall-shadow.

For example, the outlet of the radiator in an underhood application is typically of **interior** type (that is, has cells on both sides). If you know the mass flow rate through this zone (either from other CFD models or from test data) and want to apply it as a boundary condition at the radiator outlet, you first need to slit the radiator outlet. To be able to slit it, select **wall** from the **Type** drop-down list in the **Boundary Conditions** task page for this outlet. It will create a wall and a shadow. Then you can use the TUI command (`mesh/modify-zones/slit-face-zone`) to slit them. After it is slit, two additional walls will be created, one facing one side of the outlet and another facing the other. Then you can select the appropriate wall and change the **Type** to **mass-flow-inlet** and specify the mass flow rate using the Mass-Flow Inlet Dialog Box (p. 2411). There is no option of **mass-flow-inlet** without first slitting it.

Important

You should not confuse "slitting" a face zone with "separating" a face zone. When you slit a face zone, additional faces and nodes are created and placed in a new zone. When you separate a face zone, a new zone will be created, but no new faces or nodes are created; the existing faces and nodes are simply redistributed among the zones.

5.9.6.1. Inputs for Slitting Face Zones

If you want to slit a face zone, you can use the `mesh/modify-zones/slit-face-zone` text command.

`mesh` → `modify-zones` → `slit-face-zone`

You will specify the face zone's name or ID, and Fluent will replace the zone with two zones:

```
/mesh/modify-zones > slit-face-zone

 Face zone id/name [] wall-4

   zone 4 deleted
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
face zone 4 created
face zone 10 created
```

Important

Remember to save a new case file (and a data file, if data exists) after slitting faces.

5.9.7. Orienting Face Zones

The faces of a boundary face zone (which have cells only on one side) are oriented such that the normals are all pointing in one direction. However, for internal face zones (which have cells on both sides), the normals are allowed to point in either direction. To orient them so that they all point in one direction, you can use the following TUI command:

`mesh` → `modify-zones` → `orient-face-zone`

Having all of the normals oriented in one direction is needed for some boundary condition types. For example, the **fan** boundary condition type determines the flow direction based on its normals. If some of the normals are pointing in one direction and some in the other, the correct flow orientation cannot be determined, which leads to incorrect results.

5.9.8. Extruding Face Zones

The ability to extrude a boundary face zone allows you to extend the solution domain without having to exit Fluent. A typical application of the extrusion capability is to extend the solution domain when recirculating flow is impinging on a flow outlet. The current extrusion capability creates prismatic or hexahedral layers based on the shape of the face and normal vectors computed by averaging the face normals to the face zone's nodes. You can define the extrusion process by specifying a list of displacements (in SI units) or by specifying a total distance (in SI units) and parametric coordinates.

Important

- Note that extrusion is not possible from boundary face zones that have hanging nodes.
- Extruding face zones is not allowed on polygonal face zones.
- Extruding face zones is only allowed in the 3D version of ANSYS Fluent.

5.9.8.1. Specifying Extrusion by Displacement Distances

You can specify the extrusion by entering a list of displacement distances (in SI units) using the `mesh/modify-zones/extrude-face-zone-delta` text command.

`mesh` → `modify-zones` → `extrude-face-zone-delta`

Note

This text command is not available in the parallel version of ANSYS Fluent.

You will be prompted for the boundary face zone ID or name and a list of displacement distances.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.9.8.2. Specifying Extrusion by Parametric Coordinates

You can specify the extrusion by specifying a distance (in SI units) and parametric coordinates using the `mesh/modify-zones/extrude-face-zone-para` text command.

`mesh` → `modify-zones` → `extrude-face-zone-para`

Note

This text command is not available in the parallel version of ANSYS Fluent.

You will be prompted for the boundary face zone ID or name, a total distance, and a list of parametric coordinates. The list of parametric coordinates should begin with 0.0 and end with 1.0. For example, the following list of parametric coordinates would create two equally spaced extrusion layers: 0.0, 0.5, 1.0.

5.9.9. Replacing, Deleting, Deactivating, and Activating Zones

ANSYS Fluent allows you to append or replace an existing cell zone in the mesh. You can also permanently delete a cell zone and all associated face zones, or temporarily deactivate and activate zones from your ANSYS Fluent case.

5.9.9.1. Replacing Zones

This feature allows you to replace a small region of a computational domain with a new region of different mesh quality. This functionality will be required where you may want to make changes to the geometry or mesh quality for any part of the domain. This ability of ANSYS Fluent will save you time, since you can modify only the required part of the domain without remeshing the whole domain every time.

The replacement mesh must be prepared in advance, with the following considerations:

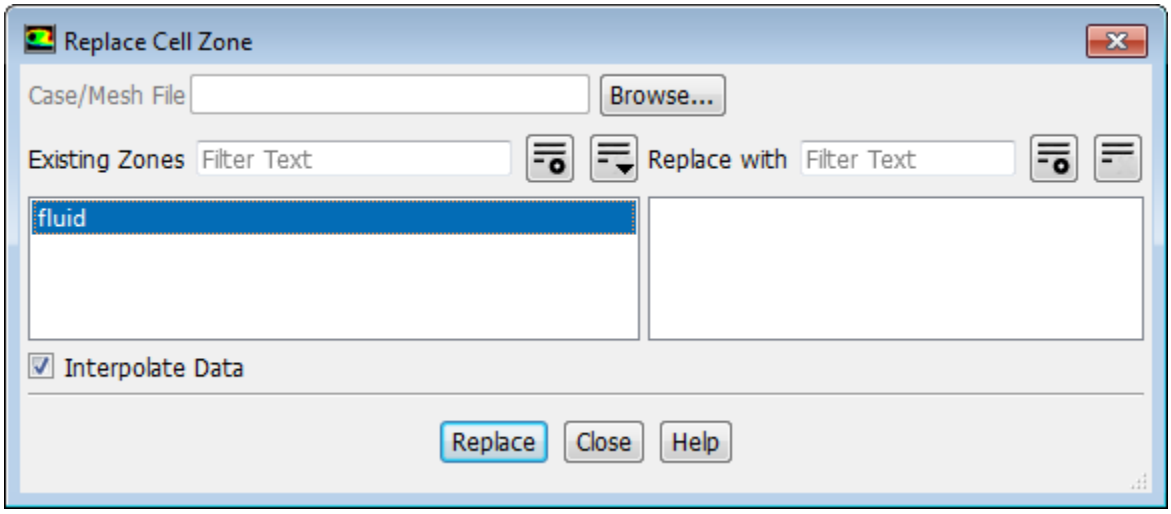
- The replacement mesh must contain a single cell zone.
- The cell zone in the replacement mesh must be of the same zone type as the zone that is being replaced.
- If the boundaries of the replacement mesh will form a non-conformal interface with existing zones (that is, zones that are not replaced in the existing mesh), these boundaries should be based on the same geometry as the existing zones. If there are sharp features (for example, 90-degree angles) or curvature in the mesh, it is especially important that both sides of the interface closely follow that feature.
- The replacement cell zone does not need to have the same name as the existing zone being replaced.
- When naming the boundary zones of the replacement cell zone, note that boundary conditions will be automatically transferred from the existing cell zone based on name. All boundaries of the replacement cell zone that have unique names will be assigned the default boundary condition for that zone type.

Replacing a zone is performed using the Replace Cell Zone Dialog Box (p. 2785) (Figure 5.71: The Replace Cell Zone Dialog Box (p. 219)).

Setting Up Domain → **Zones** → **Replace Zone...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.71: The Replace Cell Zone Dialog Box



To replace a zone, do the following:

1. Click **Browse...** and select the new or modified mesh containing the cell zone that will replace one of the cell zones in the current mesh.

   *The name of the cell zone in the replacement mesh will be displayed in the **Replace with** list.*

2. Under **Existing Zones**, select the zone you want to replace.

3. Under **Replace with**, select the zone from the replacement mesh.

4. Enable/Disable **Interpolate Data**, if data already exists. If the replacement cell zone is geometrically different, then **Interpolate Data** can be turned off to prevent data interpolation over the non-matching geometries.

   Important

   Disable **Interpolate Data** if the zone being replaced is part of an overset interface.

5. Click **Replace** to replace the selected zone.

6. Set up the boundary conditions for any of the boundaries of the replacement mesh that do not share a name with one of the existing boundaries.

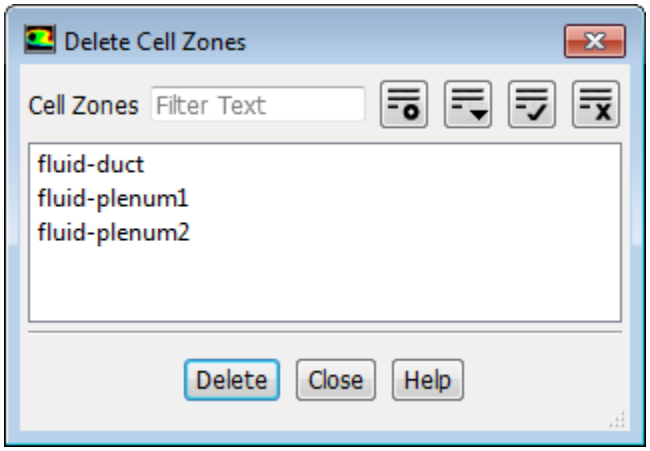
5.9.9.2. Deleting Zones

To permanently delete zones, select them in the Delete Cell Zones Dialog Box (p. 2687) (Figure 5.72: The Delete Cell Zones Dialog Box (p. 220)), and click **Delete**.

Setting Up Domain → Zones → Delete...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.72: The Delete Cell Zones Dialog Box



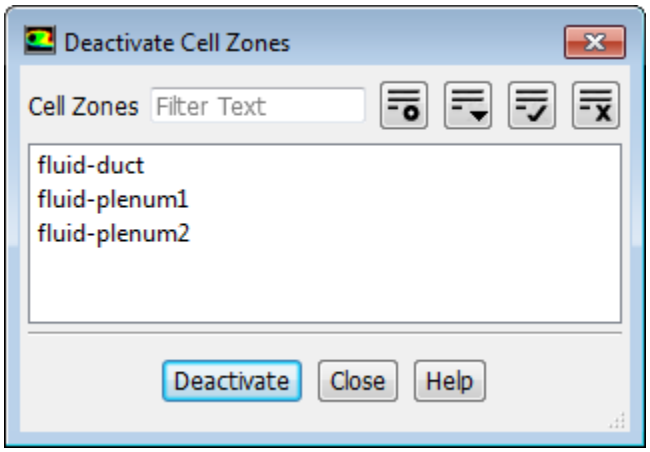
All of the cells, faces, and nodes associated with the cell zone will be deleted. If one of the faces is of type **interior** and borders another cell zone, the face will automatically be changed to a wall and will stay attached to the remaining cell zone.

5.9.9.3. Deactivating Zones

To deactivate zones, select them in the Deactivate Cell Zones Dialog Box (p. 2685) (Figure 5.73: The Deactivate Cell Zones Dialog Box (p. 220)), and click **Deactivate**.

Setting Up Domain → Zones → Deactivate...

Figure 5.73: The Deactivate Cell Zones Dialog Box



Deactivation will separate all relevant interior face zones (that is, fan, interior, porous-jump, or radiator) into wall and wall-shadow pairs.

Note

When you deactivate a zone using the **Deactivate Cell Zones** dialog box, ANSYS Fluent will remove the zone from the mesh and from all relevant solver loops.

To deactivate selected cell zones in parallel

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Read in the data file, or initialize your solution.
2. Select the zone(s) to be deactivated under **Cell Zones**.
3. Click the **Deactivate** button.

Important

If a cell zone contains mesh interfaces it cannot be deactivated until the solution has been initialized.

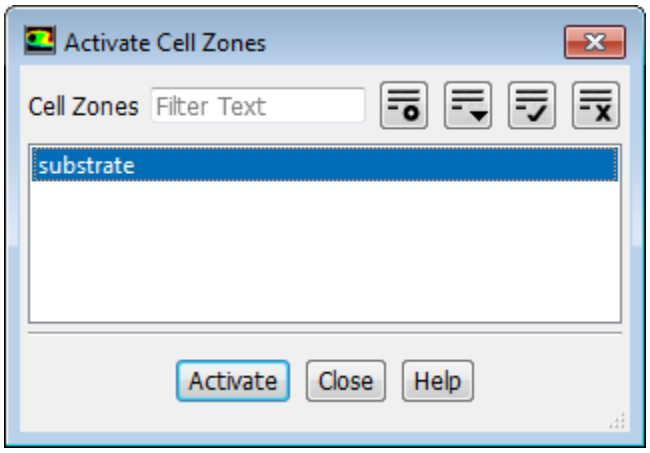
If you have neither read in the data file nor initialized the solution prior to clicking the **Deactivate** button, then the selected cell zones will only be *marked* for deactivation. The zones will not be deactivated until data is read or the solution is initialized.

5.9.9.4. Activating Zones

You can reactivate the zones and recover the last data available for them using the Activate Cell Zones Dialog Box (p. 2662) (Figure 5.74: The Activate Cell Zones Dialog Box (p. 221)).

Setting Up Domain → Zones → Activate...

Figure 5.74: The Activate Cell Zones Dialog Box



Note

The **Activate Cell Zones** dialog box will only be populated with zones that were previously deactivated.

After reactivation, you need to make sure that the boundary conditions for the wall and wall-shadow pairs are restored correctly to what you assigned before deactivating the zones. If you plan to reactivate them at a later time, make sure that the face zones that are separated during deactivation are not modified. Adaption, however, is supported.

To activate selected cell zones in parallel

1. Read in the data file, or initialize your solution.
2. Select the zone(s) to be activated under **Cell Zones**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click the **Activate** button.

Important

If you have neither read in the data file nor initialized the solution, prior to clicking the **Activate** button, then the selected cell zones will only be *marked* for activation. The zones will not be activated until data is read or the solution is initialized.

5.9.10. Copying Cell Zones

You can create a copy of a cell zone that is offset from the original either by a translational distance or a rotational angle. Note that in the copied zone, the bounding face zones are all converted to walls, any existing cell data is initialized to a constant value, and non-conformal interfaces and dynamic zones are not copied; otherwise, the model settings are the same as in the original zone.

To copy a zone, use the following text command:

`mesh` → `modify-zones` → `copy-move-cell-zone`

You will then be prompted to enter the ID of the zone you want to copy, and either the distance for translation in each of the axes or the rotational angle, origin, and axis.

Note that if you want the copied zone to be connected to existing zones, you must either fuse the boundaries (as described in Fusing Face Zones (p. 212)) or set up a non-conformal interface (as described in Using a Non-Conformal Mesh in ANSYS Fluent (p. 164)).

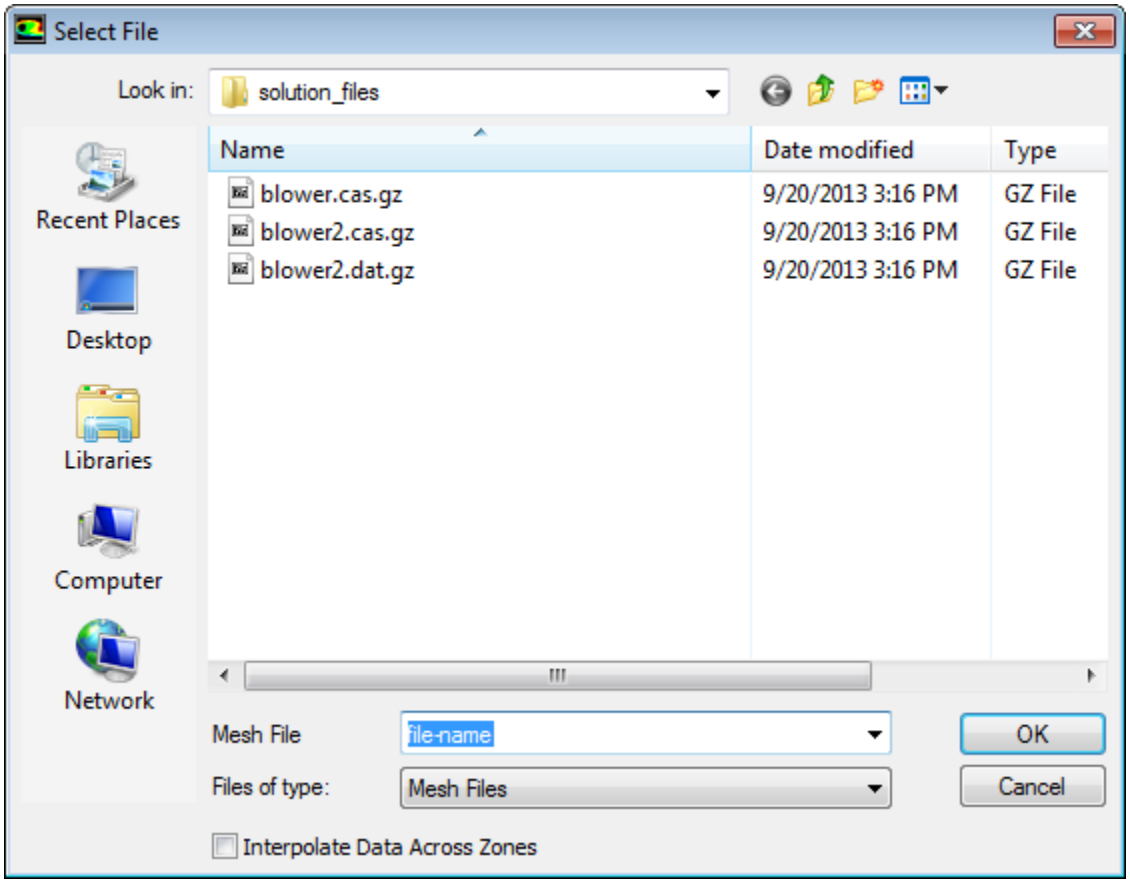
5.9.11. Replacing the Mesh

ANSYS Fluent allows you to replace your mesh, so that you can continue to run your simulation on a new mesh without having to manually copy the case file settings or interpolate the existing data. Replacing the mesh globally may be helpful, for example, if you would like to perform a mesh refinement study with a prepared series of meshes, or if your results indicate that your current mesh is not of sufficient quality. Note that the data interpolation is done automatically during the replacement if data exists (that is, you have initialized the flow field or run a calculation).

Global mesh replacement is performed by clicking **Replace Mesh...** in the **Setting Up Domain** ribbon tab **Zones** group box), which opens the **Select File** dialog box (Figure 5.75: The Select File Dialog Box (p. 223)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.75: The Select File Dialog Box



5.9.11.1. Inputs for Replacing the Mesh

The steps for replacing the mesh using the **Select File** dialog box (Figure 5.75: The Select File Dialog Box (p. 223)) are as follows:

1. Select the replacement mesh from the appropriate folder.

2. By default, data is interpolated between matching zone pairs (that is, between the zones with the same names in both the current mesh and the replacement mesh). You can enable the **Interpolate Data Across Zones** option if you want to interpolate data across cell zones when replacing the mesh. This option is appropriate when the matching zone pairs do not have the same interior zone boundaries. Note that global conservation of data is not enforced when the **Interpolate Data Across Zones** option is enabled, so it should only be used when absolutely necessary. For best data interpolation, the zone boundaries of the replacement mesh should be coincident with those of the current mesh.

   When running in parallel the replacement mesh gets partitioned such that matching cells reside on the same compute node. In some cases, differences in mesh resolution can cause the mesh to not be able to find its interpolation cell on the same compute node during interpolation. Fluent then automatically enables **Interpolate Data Across Zones** to ensure successful data interpolation across zones, even if you have not enabled the option. To avoid this behavior use the Scheme command: `(rpsetvar 'dynamesh/replace-mesh/partition-per-zone? #t)`. Note that using this command comes at the expense of requiring additional repartitioning steps.

3. Click the **OK** button to replace the mesh.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.9.11.2. Limitations

The following limitations apply when replacing meshes:

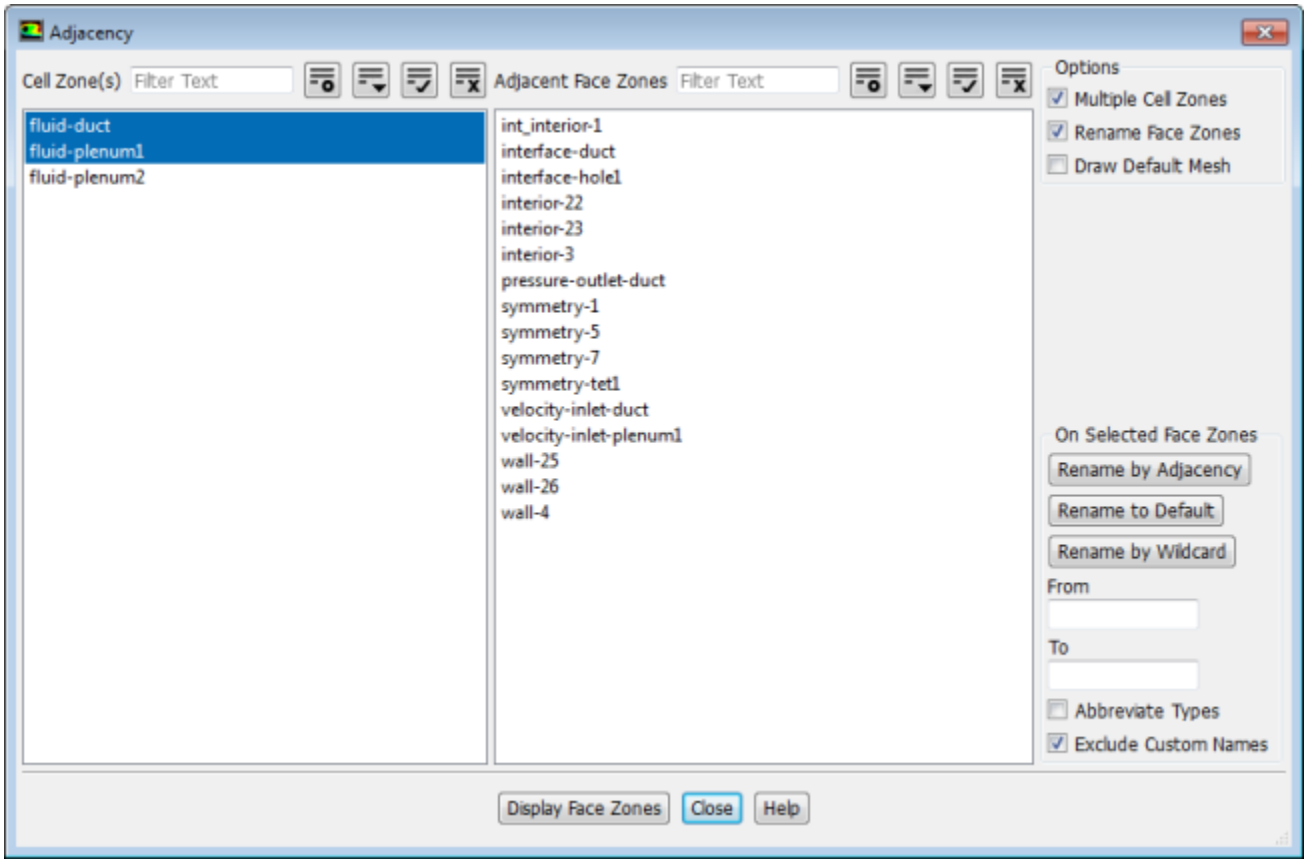
- The boundary conditions of the zones in the current mesh are mapped onto the zones with the same names in the replacement mesh, as described in Reading and Writing Boundary Conditions (p. 56). If your replacement mesh contains zones for which no match is found, these zones will have the default boundary conditions for that zone type.
- The replacement mesh is expected to be an ANSYS Fluent mesh (`.msh`) file. You can select an ANSYS Fluent case (`.cas`) file instead, but be aware that only the mesh information will be used and all of the setup information associated with that case file will be ignored.
- If you intend to select an ANSYS Fluent case (`.cas`) file as the replacement mesh, you must first delete any defined non-conformal mesh interfaces in the case file (as described in Using a Non-Conformal Mesh in ANSYS Fluent (p. 164)).
- You should ensure that the replacement mesh has the same mesh scaling as your current mesh. The data interpolation will not work properly if the meshes are scaled differently.

5.9.12. Managing Adjacent Zones

In some cases you may want to identify and display the face zones which are adjacent to a cell zone. You can do this using the **Adjacency** dialog box.

Setting Up Domain → Zones → Adjacency...

Figure 5.76: The Adjacency Dialog Box



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using the **Adjacency** dialog box (Figure 5.76: The Adjacency Dialog Box (p. 224)) you can also optionally rename zones based on their adjacency or with default names based on zone type. The steps to use the **Adjacency** dialog box are:

1. Open the **Adjacency** dialog box by clicking **Adjacency...** in the **Setting Up Domain** ribbon tab (**Zones** group box).

2. Select a zone under **Cell Zone(s)** to populate the **Adjacent Face Zones** list. You can use the **Cell Zone Types** list box to select or deselect cell zones by type. If the **Multiple Cell Zones** option is disabled, only one cell zone can be selected at a time.

3. Select the face zones you want to display in **Adjacent Face Zones** and click the **Display Face Zones** button.

4. Enable **Draw Default Mesh** to open the **Mesh Display** dialog box where you may choose to display mesh zones. These will be displayed permanently while others will be displayed as currently selected in the **Adjacent Face Zones** list. This is useful for finding your way through a new and complex mesh.

5. Enable **Rename Face Zones** if you want to rename or clean up some of the names that may cause confusion. See Renaming Zones Using the Adjacency Dialog Box (p. 225) for more information.

5.9.12.1. Renaming Zones Using the Adjacency Dialog Box

You can rename selected zones from the Adjacency dialog box after enabling **Rename Face Zones**. The following renaming methods are available:

Rename by Adjacency
appends the name of the cell zone to the type of the adjacent face zone. For example, if **fluid** is selected in the **Cell Zone(s)** and **interior** is selected in the **Adjacent Face Zones** list, then renaming by adjacency produces **interior-fluid**. If the name is already in use, the original zone ID is appended in addition in order to create a unique name. Note that it is best if you avoid using long cell zone names.

Rename to Default
simply appends the original zone ID to the zone type. Enable **Abbreviate Types** to abbreviate the zone type (for example, `vi` for velocity inlet, `int` for interior, `ifc` for interface, and so on).

Rename by Wildcard
performs pattern matching of the selected zone names against the pattern in the **From** text box (with optional wildcards) and replaces a matching substring with the literal string in the **To** text box. Wildcard characters * and ? are interpreted as follows:

- A * at the beginning and/or end of the **From** pattern matches zero or more characters at the beginning and/or end of the zone name that are retained during renaming with the rest of the name replaced by the **To** string.

  A * used anywhere except at the beginning or end of the **From** pattern is not supported and may have unpredictable results.

- A ? matches exactly one character and is considered part of the string to be replaced. Multiple ? characters can be used to match substrings of more than one character.

For all of the renaming methods, you can enable **Exclude Custom Names** to exclude customized names (those that do not conform to a recognized pattern or match any default names) from renaming. This is a protective measure so as not to accidentally destroy your desired naming. Disabling this option will

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

unconditionally rename all selected zones according to the specified renaming method and customized names will be permanently lost.

5.9.13. Reordering the Domain

In a Fluent session that uses multiple processes, the mesh domain is automatically reordered when it is read, in order to increase memory access efficiency. The reordering uses the Reverse Cuthill-McKee algorithm [24] (p. 2888), and is applied to cells and faces.

A typical output produced during the domain reordering at read is shown below:

```
Building...
     mesh
 auto partitioning mesh by Metis (fast),
 distributing mesh
  parts...,
  faces...,
  nodes...,
  cells...,
        bandwidth reduction using Reverse Cuthill-McKee: 3875/150 = 25.8333
```

The bandwidth is the maximum difference between neighboring cells in the zone—that is, if you numbered each cell in the zone list sequentially and compared the differences between these indices.

5.9.14. Scaling the Mesh

Internally, ANSYS Fluent stores the computational mesh in meters, the SI unit of length. When mesh information is read, it is assumed that the mesh was generated in units of meters. If your mesh was created using a different unit of length (inches, feet, centimeters, and so on), you must scale the mesh to meters. To do this, you can select from a list of common units to convert the mesh or you can supply your own custom scale factors. Each node coordinate will be multiplied by the corresponding scale factor.

Scaling can also be used to change the physical size of the mesh. For instance, you could stretch the mesh in the x direction by assigning a scale factor of 2 in the x direction and 1 in the y and z directions. This would double the extent of the mesh in the x direction. However, you should use anisotropic scaling with caution, since it will change the aspect ratios of the cells in your mesh.

Important

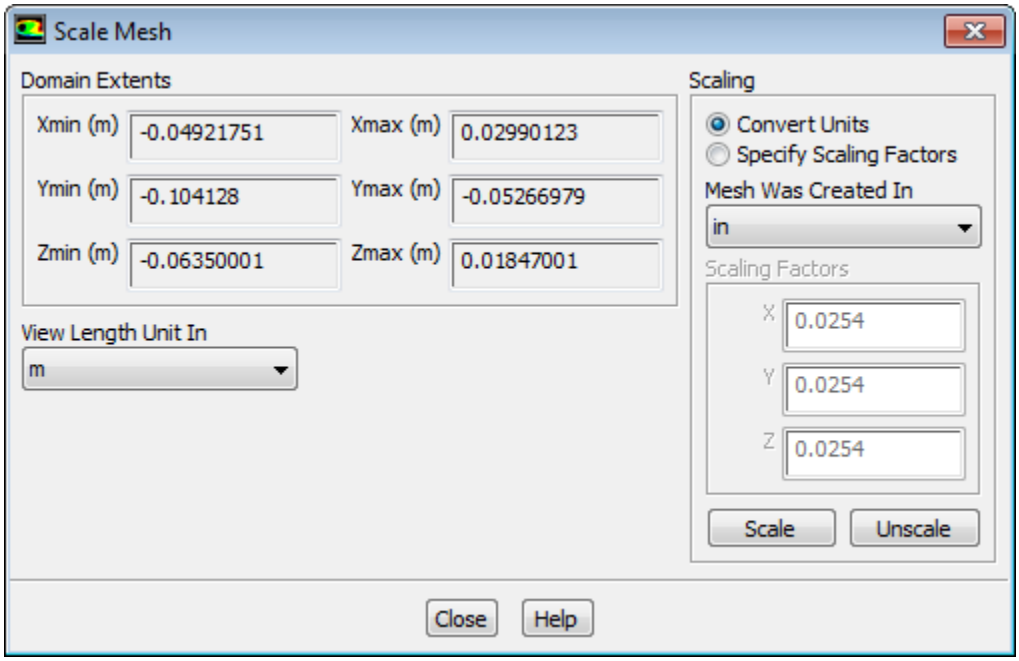
- If you plan to scale the mesh in any way, you should do so before you compute the view factors (as part of an S2S radiation problem), initialize the flow, or begin calculations. Any data that exists when you scale the mesh will be invalid.
- It is a good practice to scale the mesh before setting up the case, especially when you plan to create mesh interfaces or shell conduction zones.

You will use the Scale Mesh Dialog Box (p. 2162) (Figure 5.77: The Scale Mesh Dialog Box (p. 227)) to scale the mesh from a standard unit of measurement or to apply custom scaling factors.

Setup → General → Scale...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.77: The Scale Mesh Dialog Box



5.9.14.1. Using the Scale Mesh Dialog Box

The procedure for scaling the mesh is as follows:

1. Use the conversion factors provided by ANSYS Fluent by selecting **Convert Units** in the **Scaling** group box. Then indicate the units used when creating the mesh by selecting the appropriate abbreviation for meters, centimeters, millimeters, inches, or feet from the **Mesh Was Created In** drop-down list. The **Scaling Factors** will automatically be set to the correct values (for example, 0.0254 meters/inch).

   If you created your mesh using units other than those in the **Mesh Was Created In** drop-down list, you can select **Specify Scaling Factors** and enter values for **X**, **Y**, and **Z** manually in the **Scaling Factors** group box (for example, the number of meters per yard).

2. Click the **Scale** button. The **Domain Extents** will be updated to show the correct range in meters. If you prefer to use your original unit of length during the ANSYS Fluent session, you can follow the procedure described below to change the unit.

5.9.14.1.1. Changing the Unit of Length

As mentioned in Step 2. of the previous section, when you scale the mesh you do not change the units; you just convert the original dimensions of your mesh points from your original units to meters by multiplying each node coordinate by the specified **Scaling Factors**. If you want to work in your original units, instead of in meters, you can make a selection from the **View Length Unit In** drop-down list. This updates the **Domain Extents** to show the range in your original units and automatically changes the length unit in the Set Units Dialog Box (p. 2166) (see Customizing Units (p. 112)). Note that this unit will be used for all future inputs of length quantities.

5.9.14.1.2. Unscaling the Mesh

If you use the wrong scale factor, accidentally click the **Scale** button twice, or want to undo the scaling for any other reason, you can click the **Unscale** button. "Unscaling" simply divides each of the node

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

coordinates by the specified **Scale Factors**. (Selecting **m** in the **Mesh Was Created In** list and clicking on **Scale** will *not* unscale the mesh.)

5.9.14.1.3. Changing the Physical Size of the Mesh

You can also use the Scale Mesh Dialog Box (p. 2162) to change the physical size of the mesh. For example, if your 2D mesh is 5 feet by 8 feet, and you want to model the same geometry with dimensions twice as big (10 feet by 16 feet), you can enter 2 for **X** and **Y** in the **Scaling Factors** group box and click **Scale**. The **Domain Extents** will be updated to show the new range.

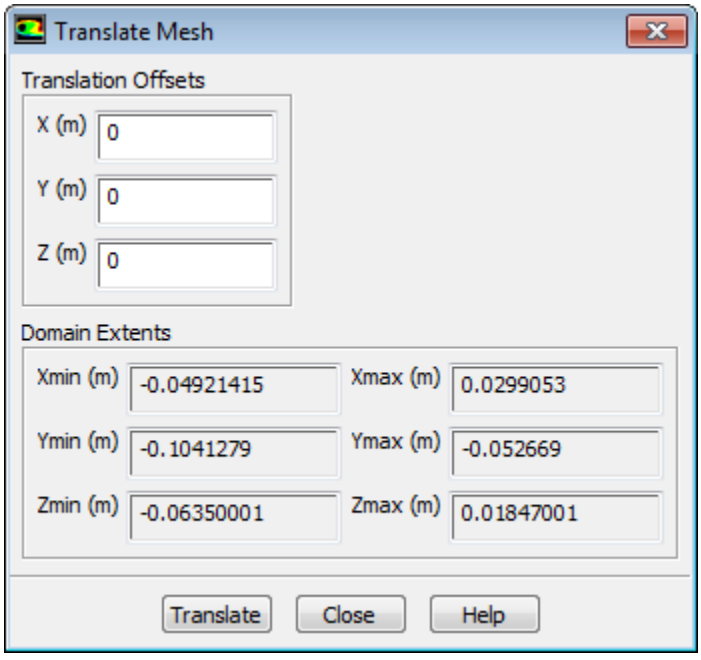
5.9.15. Translating the Mesh

You can "move" the mesh by applying prescribed offsets to the Cartesian coordinates of all the nodes in the mesh. This would be necessary for a rotating problem if the mesh were set up with the axis of rotation not passing through the origin, or for an axisymmetric problem if the mesh were set up with the axis of rotation not coinciding with the x axis. It is also useful if, for example, you want to move the origin to a particular point on an object (such as the leading edge of a flat plate) to make an XY plot have the desired distances on the x axis.

You can translate mesh points in ANSYS Fluent using the Translate Mesh Dialog Box (p. 2813) (Figure 5.78: The Translate Mesh Dialog Box (p. 228)).

Setting Up Domain → Mesh → Transform → Translate...

Figure 5.78: The Translate Mesh Dialog Box



5.9.15.1. Using the Translate Mesh Dialog Box

The procedure for translating the mesh is as follows:

1. Enter the desired translations in the x, y, and (for 3D) z directions (that is, the desired delta in the axes) in the **X**, **Y**, and **Z** text-entry boxes in the **Translation Offsets** group box. You can specify positive or negative real numbers in the current unit of length.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click the **Translate** button and redisplay the mesh. The **Domain Extents** will be updated to display the new extents of the translated mesh. (Note that the **Domain Extents** are purely informational; you cannot edit them manually.)

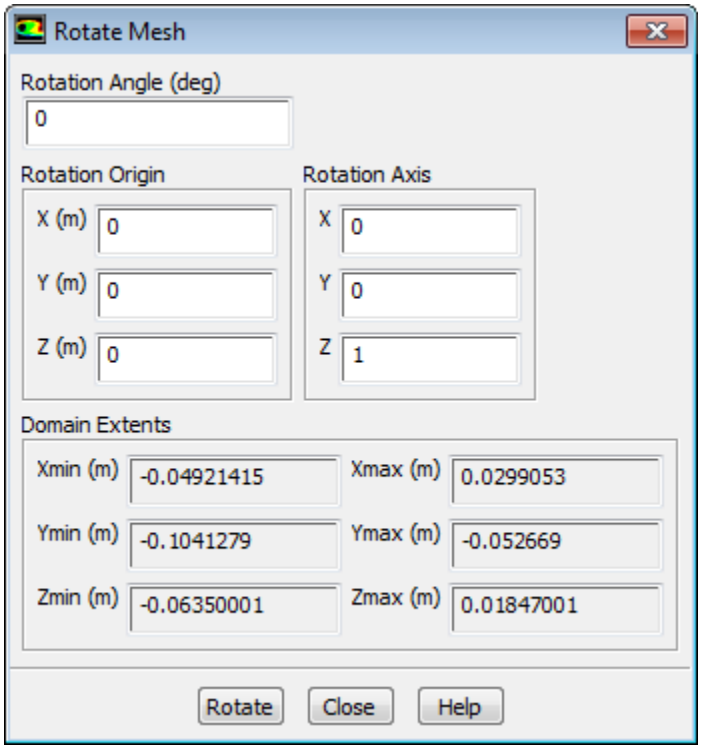
5.9.16. Rotating the Mesh

The ability to rotate the mesh is analogous to the ability to translate the mesh in ANSYS Fluent. You can rotate the mesh about the x, y, or (for 3D) z axis and also specify the rotation origin. This option is useful in the cases where the structural mesh and the CFD mesh are offset by a small angle.

You can rotate the mesh in ANSYS Fluent using the Rotate Mesh Dialog Box (p. 2792) (Figure 5.79: The Rotate Mesh Dialog Box (p. 229)).

Setting Up Domain → Mesh → Transform → Rotate...

Figure 5.79: The Rotate Mesh Dialog Box



5.9.16.1. Using the Rotate Mesh Dialog Box

The procedure for rotating the mesh is as follows:

1. Specify the required **Rotation Angle** for the mesh. You can specify any positive or negative real number in the correct unit of angle.

2. In the **Rotation Origin** group box, enter **X**, **Y**, and (for 3D) **Z** coordinates to specify a new origin for the axis of rotation.

3. In the **Rotation Axis** group box, enter values for the **X**, **Y**, and (for 3D) **Z** axes to specify the vector for the axis of rotation.

4. Click the **Rotate** button and redisplay the mesh.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Domain Extents will be updated to display the new extents of the rotated mesh. (Note that the Domain Extents are purely informational; you cannot edit them manually.)

5.9.17. Improving the Mesh by Smoothing and Swapping

Smoothing and face swapping are tools that complement mesh adaption by increasing the quality of the final numerical mesh. Smoothing repositions the nodes, and face swapping modifies the cell connectivity to achieve these improvements in quality.

Important

Only smoothing that is based on quality is recommended; face swapping and smoothing based on skewness or a Laplacian operator have more limitations, and support for them may be removed in a future release.

Important

Face swapping is applicable only to meshes with triangular or tetrahedral cells.

Important

Face swapping and smoothing based on skewness or a Laplacian operator are available only for serial cases; only quality-based smoothing can be used for parallel cases.

For additional information, see the following sections:

5.9.17.1. Smoothing
5.9.17.2. Face Swapping
5.9.17.3. Combining Skewness-Based Smoothing and Face Swapping

5.9.17.1. Smoothing

The three smoothing methods that are available in ANSYS Fluent are:

- quality-based smoothing

  This method is the only recommended method, and is available for all types of meshes in serial and parallel.

- Laplacian smoothing

  This method is not recommended, though it is available for all types of meshes in serial (and is best suited for quadrilateral and hexahedral meshes).

- skewness-based smoothing

  This method is not recommended, though it is available for triangular and tetrahedral meshes in serial; it can be used alternatively with face swapping (see Combining Skewness-Based Smoothing and Face Swapping (p. 236)).

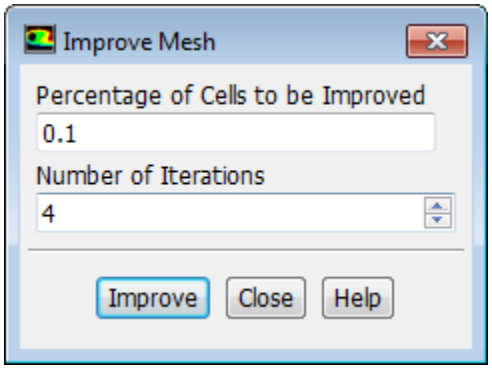
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.9.17.1.1. Quality-Based Smoothing

Smoothing that is based on quality is performed using the Improve Mesh Dialog Box (p. 2715) (Figure 5.80: The Improve Mesh Dialog Box (p. 231)).

Setting Up Domain → Mesh → Improve...

Figure 5.80: The Improve Mesh Dialog Box



ANSYS Fluent divides the mesh into a number of "bins", each of which contain a certain number of cells. Improvements are attempted on the cells in those bins that exhibit the lowest orthogonal quality (as defined in Mesh Quality (p. 131)). As part of this method, you specify the percentage of the total number of cells, in order to determine how many bins are modified.

Note that this smoothing is similar to when you use the `mesh/repair-improve/improve` text command. The advantage of using the **Improve Mesh** dialog box rather than the text command is that you can control the percentage of the cells that ANSYS Fluent attempts to improve.

To perform quality-based smoothing, do the following steps:

1. In the Improve Mesh Dialog Box (p. 2715) (Figure 5.80: The Improve Mesh Dialog Box (p. 231)), enter the **Percentage of Cells to be Improved**.

   Important

   Quality-based smoothing will be CPU intensive if you specify a large value for the **Percentage of Cells to be Improved**. It is recommended that you enter a small value initially, and then perform the smoothing process multiple times, if necessary. Note that the maximum percentage allowed is 10%.

2. Specify the number of successive smoothing sweeps to be performed on the mesh in the **Number of Iterations** number-entry box. The default value is 4.

3. Click the **Improve** button.

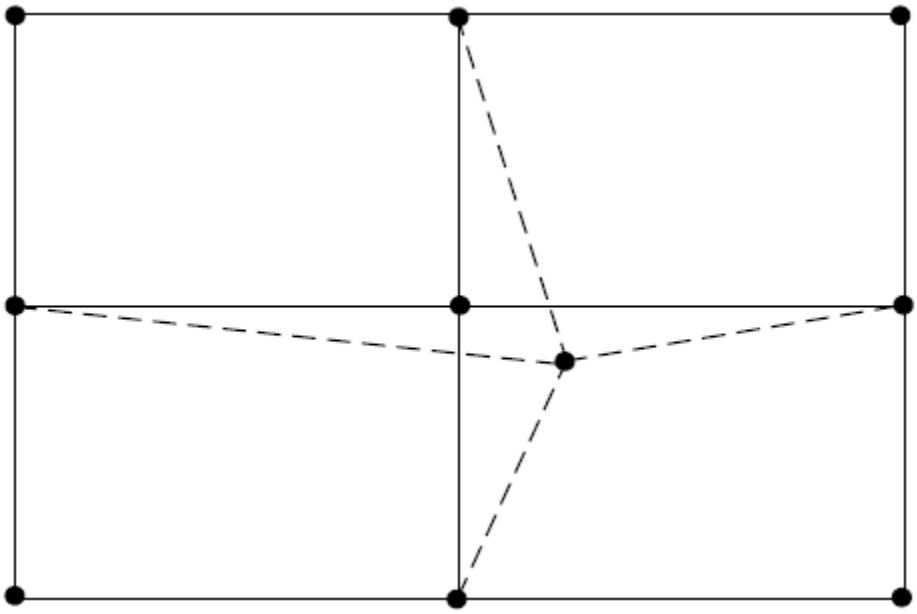
5.9.17.1.2. Laplacian Smoothing

When you use this method, a Laplacian smoothing operator is applied to the unstructured mesh to reposition nodes. The new node position is the average of the positions of its node neighbors. The computed node position increment is multiplied by the relaxation factor (which is set to a value between 0.0 and 1.0). A value of zero for the relaxation factor results in no movement of the node, and a value

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of unity results in movement equivalent to the entire computed increment. Figure 5.81: Result of Smoothing Operator on Node Position (p. 232) illustrates the new node position for a typical configuration of quadrilateral cells. The dashed line is the original mesh and the solid line is the final mesh.

Figure 5.81: Result of Smoothing Operator on Node Position



This repositioning strategy improves the skewness of the mesh, but relaxes the clustering of node points. In extreme circumstances, the present operator may create mesh lines that cross over the boundary, creating negative cell volumes. This is most likely to occur near sharp or coarsely resolved convex corners, especially if you perform multiple smoothing operations with a large relaxation factor. Figure 5.82: Initial Mesh Before Smoothing Operation (p. 233) illustrates an initial tetrahedral mesh before one unrelaxed smoothing iteration creates mesh lines that cross over each other (Figure 5.83: Mesh Smoothing Causing Mesh-Line Crossing (p. 233)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.82: Initial Mesh Before Smoothing Operation

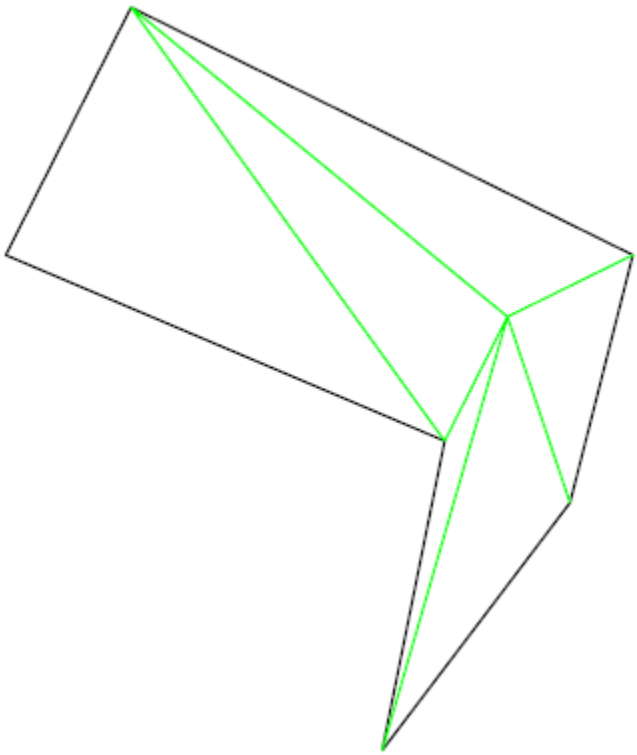
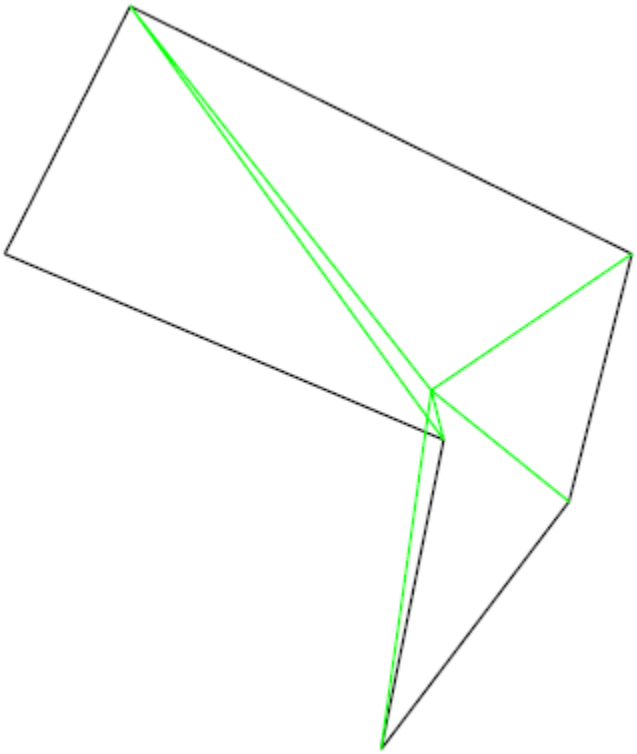


Figure 5.83: Mesh Smoothing Causing Mesh-Line Crossing



The default smoothing parameters are designed to improve mesh quality with minimal adverse effects, but it is recommended that you save a case file before smoothing the mesh. If you apply a conservative relaxation factor and start with a good quality initial mesh, the frequency of failure due to smoothing

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

is extremely low in two dimensions. However, corruption of the mesh topology occurs much more frequently in three dimensions, particularly with tetrahedral meshes.

The smoothing operator can also be applied repeatedly, but as the number of smoothing sweeps increase, the node points have a tendency to pull away from boundaries and the mesh tends to lose any clustering characteristics.

To perform Laplacian smoothing, enter the `mesh/smooth-mesh` text command in the console and respond to the prompts as follows:

1. Enter `"laplace"` for the `type of smoothing`.
2. Specify the number of successive smoothing sweeps to be performed on the mesh in response to the `number of iterations` prompt. The default value is 4.
3. Specify the factor by which to multiply the computed position increment for the node in response to the `relaxation factor` prompt. The lower the factor, the more reduction in node movement.

5.9.17.1.3. Skewness-Based Smoothing

When you use skewness-based smoothing, ANSYS Fluent applies a smoothing operator to the mesh, repositioning interior nodes to lower the maximum skewness of the mesh. ANSYS Fluent will try to move interior nodes to improve the skewness of cells with skewness greater than the specified "skewness threshold". This process can be very time consuming, so perform smoothing only on cells with high skewness.

Improved results can be obtained by smoothing the nodes several times. There are internal checks that will prevent a node from being moved if moving it causes the maximum skewness to increase, but it is common for the skewness of some cells to increase when a cell with a higher skewness is being improved. Thus, you may see the average skewness increase while the maximum skewness is decreasing.

Important

Carefully consider whether the improvements to the mesh due to a decrease in the maximum skewness are worth the potential increase in the average skewness. Performing smoothing only on cells with very high skewness (for example, 0.8 or 0.9) may reduce the adverse effects on the average skewness.

To perform skewness-based smoothing, enter the `mesh/smooth-mesh` text command in the console and respond to the prompts as follows:

1. Enter `"skewness"` for the `type of smoothing`.
2. Specify the number of successive smoothing sweeps to be performed on the mesh in response to the `number of iterations` prompt. The default value is 4.
3. Specify the minimum cell skewness value for which node smoothing will be attempted in response to the `skewness threshold` prompt. ANSYS Fluent will try to move interior nodes to improve the skewness of cells with skewness greater than this value. By default, skewness threshold is set to 0.4 for 2D cases and 0.8 for 3D cases.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.9.17.2. Face Swapping

While it is not recommended over smoothing (and support for it may be removed in future releases), face swapping can be used to improve the quality of a triangular or tetrahedral mesh in the serial version of ANSYS Fluent.

To perform face swapping, enter the `mesh/swap-mesh-faces` text command repeatedly until the console report declares that 0 faces were swapped out. The console report will display the total number of faces that were visited and tested for possible face swapping.

5.9.17.2.1. Triangular Meshes

The approach for triangular meshes is to use the Delaunay circle test to decide if a face shared by two triangular cells should be swapped. A pair of cells sharing a face satisfies the circle test if the circumcircle of one cell does not contain the unshared node of the second cell. Figure 5.84: Examples of Cell Configurations in the Circle Test (p. 235) illustrates cell neighbors in the circle test. In cases where the circle test is not satisfied, the diagonal or face is swapped, as illustrated in Figure 5.85: Swapped Faces to Satisfy the Delaunay Circle Test (p. 235).

Figure 5.84: Examples of Cell Configurations in the Circle Test

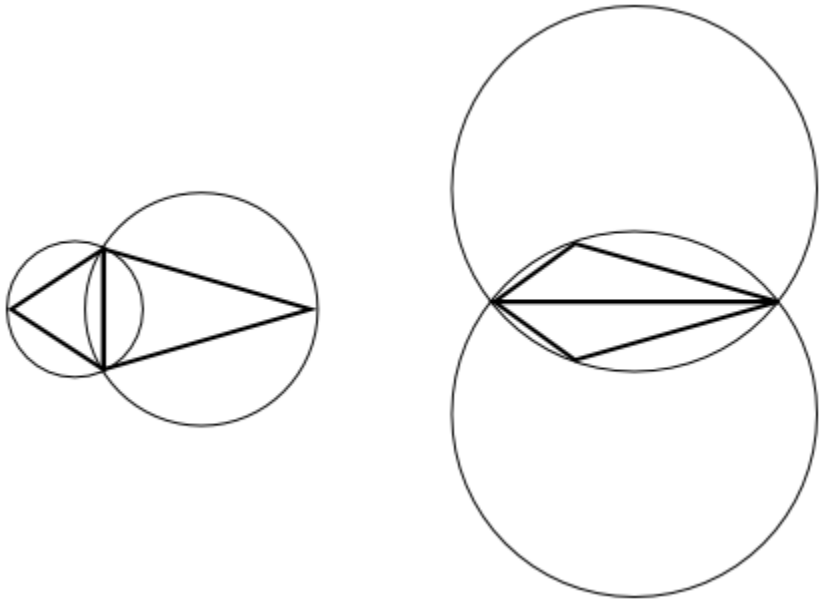


Figure 5.85: Swapped Faces to Satisfy the Delaunay Circle Test

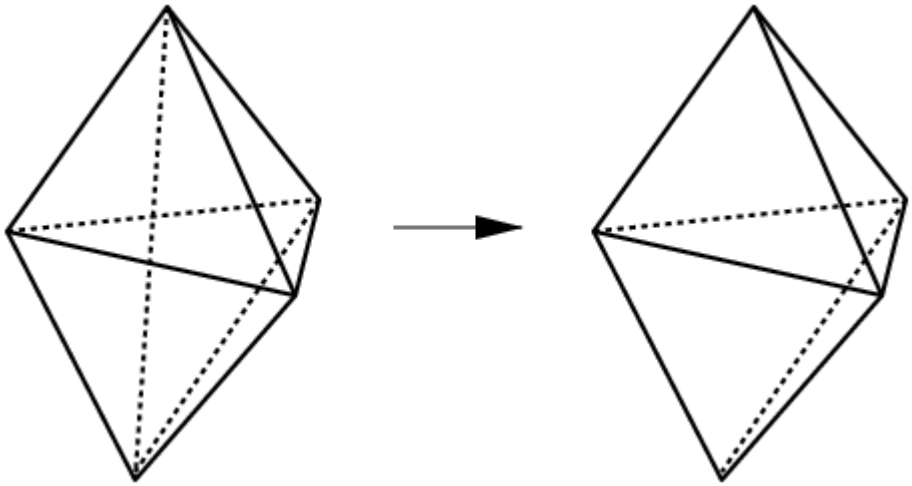
Repeated application of the face-swapping technique will produce a constrained Delaunay mesh. If you have a Delaunay mesh, it is a unique triangulation that maximizes the minimum angles in the mesh. Thus, the triangulation tends toward equilateral cells, providing the most equilateral mesh for the given node distribution. For more information on Delaunay mesh generation, see Generating Tetrahedral Meshes in the Fluent Meshing User's Guide.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.9.17.2.2. Tetrahedral Meshes

For tetrahedral meshes, face swapping consists of searching for configurations of three cells sharing an edge and converting them into two cells sharing a face to decrease skewness and the cell count (see Figure 5.86: 3D Face Swapping (p. 236)).

Figure 5.86: 3D Face Swapping



5.9.17.3. Combining Skewness-Based Smoothing and Face Swapping

As mentioned in Skewness-Based Smoothing (p. 234), skewness-based smoothing should usually be alternated with face swapping. Guidelines for this procedure are presented here.

- Perform four smoothing iterations using a `skewness threshold` of 0.8 for 3D cases, or 0.4 for 2D cases.
- Swap until the number of faces swapped decreases to 0.
- For 3D meshes, decrease the `skewness threshold` to 0.6 and repeat the smoothing/swapping procedure.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 6: Cell Zone and Boundary Conditions

This chapter describes the cell zone and boundary condition options available in ANSYS Fluent. Details regarding the cell zone and boundary condition inputs and the internal treatment at boundaries are provided.

The information in this chapter is divided into the following sections:

6.1. Overview
6.2. Cell Zone Conditions
6.3. Boundary Conditions
6.4. Boundary Acoustic Wave Models
6.5. User-Defined Fan Model
6.6. Profiles
6.7. Coupling Boundary Conditions with GT-POWER
6.8. Coupling Boundary Conditions with WAVE

6.1. Overview

Cell zone and boundary conditions specify the flow and thermal variables on the boundaries of your physical model. They are, therefore, a critical component of your ANSYS Fluent simulations and it is important that they are specified appropriately.

In this chapter, most of the cell zones and boundary conditions will be described in detail, and an explanation of how to set them and where they are most appropriately used will be provided.

6.1.1. Available Cell Zone and Boundary Types
6.1.2. The Cell Zone and Boundary Conditions Task Pages
6.1.3. Changing Cell and Boundary Zone Types
6.1.4. Setting Cell Zone and Boundary Conditions
6.1.5. Copying Cell Zone and Boundary Conditions
6.1.6. Changing Cell or Boundary Zone Names
6.1.7. Defining Non-Uniform Cell Zone and Boundary Conditions
6.1.8. Defining and Viewing Parameters
6.1.9. Selecting Cell or Boundary Zones in the Graphics Display
6.1.10. Operating and Periodic Conditions
6.1.11. Highlighting Selected Boundary Zones
6.1.12. Saving and Reusing Cell Zone and Boundary Conditions

6.1.1. Available Cell Zone and Boundary Types

The cell zone and boundary zone types available in ANSYS Fluent are classified as follows:

Table 6.1: Zone Types by Category

Category	Zone Types
External	axis, degassing, exhaust fan, inlet vent, intake fan, interface, mass-flow inlet, mass-flow outlet, outflow, outlet vent, overset, pressure far-field,

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Zone Types
	pressure inlet, pressure outlet, symmetry, velocity inlet, and wall (one-sided only)
Internal	fan, interior, porous jump, radiator, RANS/LES interface, and wall (two-sided only)
Periodic	periodic
Cells	fluid, solid (porous media and 3D fans are a type of fluid cell)

Note that internal boundary zones are connected with the cells on both of their faces, whereas external boundary zones are connected with the cells on only one of their faces.

6.1.2. The Cell Zone and Boundary Conditions Task Pages

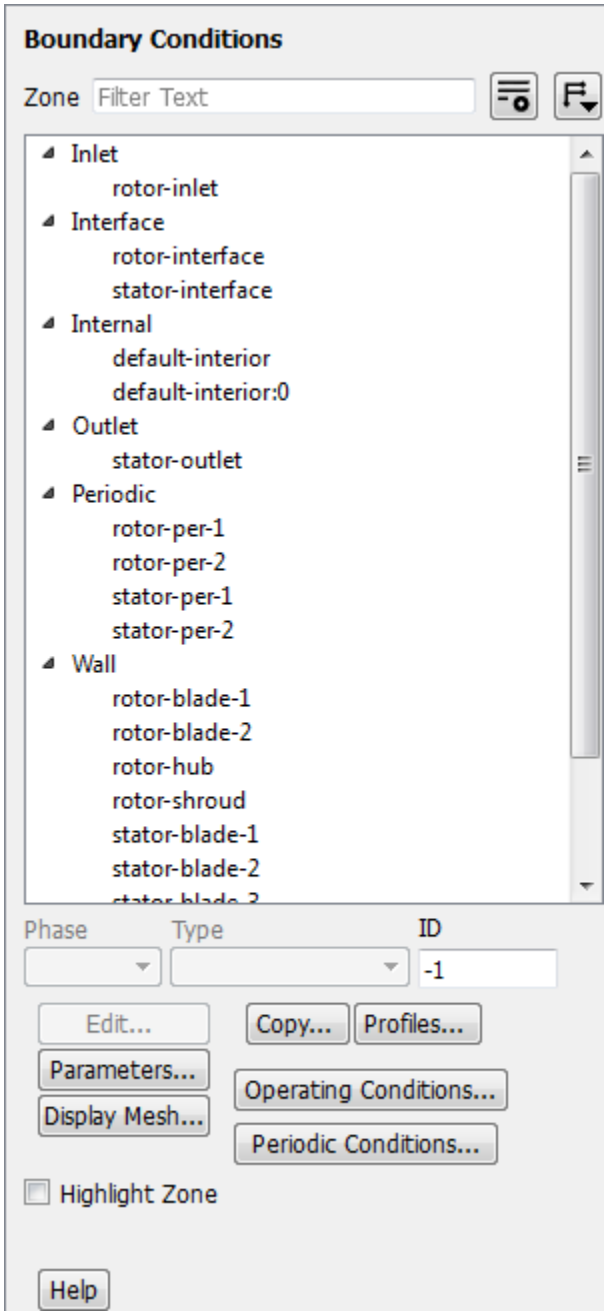
The **Cell Zone Conditions** and **Boundary Conditions** task pages (Figure 6.1: The Boundary Conditions Task Page (p. 239)) display lists of the zones (which can be grouped by name or zone type), and allow you to revise their settings and perform other related actions.

Setup → **Cell Zone Conditions**

Setup → **Boundary Conditions**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.1: The Boundary Conditions Task Page



The sections that follow explain how to perform operations with the **Cell Zone Conditions** or **Boundary Conditions** task page, and how to use the mouse and the graphics display in conjunction with the dialog box.

Note — You can also modify individual boundary conditions and cell zones by expanding the **Cell Zones** and **Boundary Conditions** branches in the tree, and right-clicking the boundary condition or cell zone you want to modify.

6.1.3. Changing Cell and Boundary Zone Types

Before you set any cell zone or boundary conditions, you should check the zone types of all boundary zones and change any if necessary. For example, if your mesh includes a pressure inlet, but you want to use a velocity inlet instead, you will need to change the pressure-inlet zone to a velocity-inlet zone.

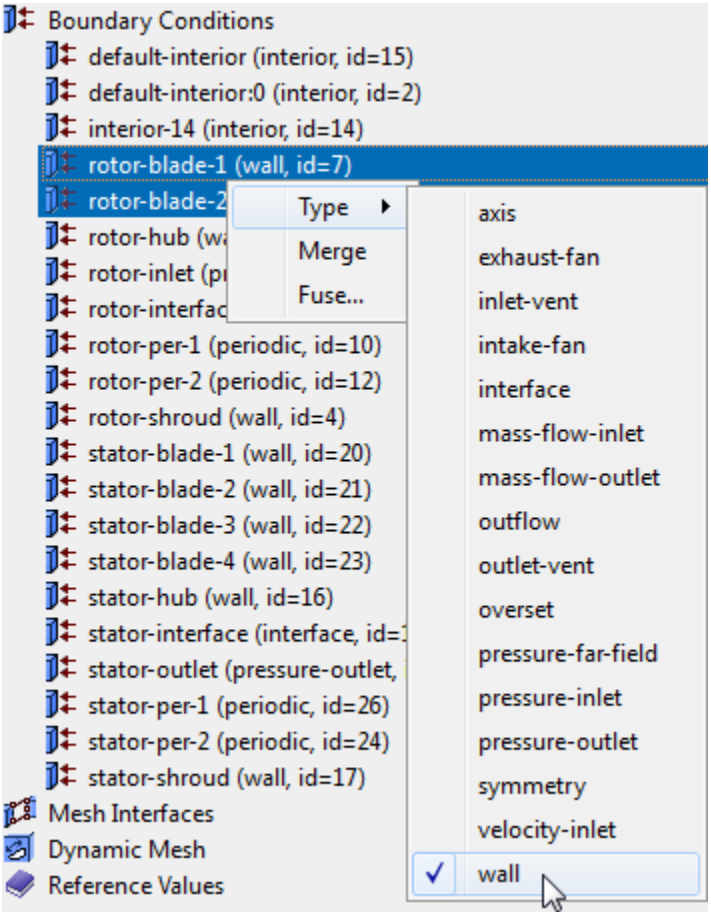
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that external and internal boundary zones (as listed in Table 6.1: Zone Types by Category (p. 237)) can only be changed to a zone of the same category.

The steps for changing a zone type using the graphical user interface are as follows:

1. Select the boundary or cell zone in the tree (under **Setup**/**Boundary Conditions** or **Setup**/**Cell Zone Conditions**). Note that you can select multiple zones by holding the **Shift** or **Ctrl** keys when selecting, though you must ensure that all of the selections are of the same category.

2. Right-click a selected boundary or cell zone, and select the correct zone type from the **Type** sub-menu.



You can also change the zone type using either the `define/boundary-conditions/zone-type` or the `define/boundary-conditions/modify-zones/zone-type` text command. To change the type for multiple zones of the same category through a single action, you can enter a list of names or IDs contained within a pair of parentheses at the prompt, as shown in the following example (which changes zone 4 and 9 to be pressure inlets):

```
> define/boundary-conditions/zone-type
zone id/name/list [] (4 9)

external-type> pressure-inlet
```

When changing the cell or boundary zone type, note the following:

- For zones that have the default name (that is, <*type*> **-** <*ID*>), the name will be changed automatically along with the type.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When you have a large number of internal boundary zones that you want to change to or from a wall / shadow pair, it is recommended that you change their type with a single action (either by multi-selecting in the tree or by using parentheses at the text command prompt); this will greatly reduce the processing time associated with the change.
- When changing a wall / shadow pair to one of the other internal types (fan, interior, porous jump, radiator, or RANS/LES interface), you can select either the wall or the wall-shadow to change. In either case, the resulting zone will retain the ID and orientation of the wall zone and the wall-shadow zone will be deleted.
- Note that you cannot use this method to change zone types to or from the periodic type, since additional restrictions exist for this boundary type. Creating Conformal Periodic Zones (p. 214) explains how to create and uncouple periodic zones.
- If you are using one of the general multiphase models (VOF, mixture, or Eulerian), the procedure for changing types is slightly different. See Steps for Setting Boundary Conditions (p. 1391) for details.

6.1.4. Setting Cell Zone and Boundary Conditions

In ANSYS Fluent, boundary conditions are associated with zones, not with individual faces or cells. If you want to combine two or more zones that will have the same boundary conditions, see Merging Zones (p. 206) for information about merging zones.

To set cell zone and boundary conditions for a particular zone, perform one of the following sequences:

1. Right-click the name of zone in the tree (under **Setup**/**Boundary Conditions** or **Setup**/**Cell Zone Conditions**).
2. Select **Edit...**.

or

1. Select the zone from the **Zone** list in the **Cell Zone Conditions** or **Boundary Conditions** task page.
2. Click the **Edit...** button.

The dialog box for the selected cell or boundary zone will open, and you can specify the appropriate conditions.

Important

If you are using one of the general multiphase models (VOF, mixture, or Eulerian), the procedure for setting conditions is slightly different from that described above. See Steps for Setting Boundary Conditions (p. 1391) for details.

You can use the text command: `define/boundary-conditions/set/` to define one or more settings at a single or multiple boundaries/cell zones of a given type. This facilitates quick and efficient setup of cell/boundary conditions for cases with large numbers of zones.

To edit multiple boundaries you can either provide a list of boundary names/ids enclosed in parenthesis:

```
/define/boundary-conditions/set> wall
(rotor-hub rotor-shroud rotor-blade-1 rotor-blade-2 stator-hub stator-shroud stator-blade-1 stator-blade-2)
zone(s) to set boundary conditions(1) [rotor-hub] (rotor-blade-1 rotor-blade-2)
```

or you can use a wildcard pattern for the boundary names:

```
/define/boundary-conditions/set> wall
(rotor-hub rotor-shroud rotor-blade-1 rotor-blade-2 stator-hub stator-shroud stator-blade-1 stator-blade-2)
zone(s) to set boundary conditions(1) [rotor-hub] (rotor-blade-*)
```

For additional information on formatting lists in the console, see *Lists (p. 32)*.

After entering the zone(s) you want to change you will be prompted for what setting to change. Pressing **Enter** will list the available settings.

When you have changed all of the desired settings, press **q** to exit the `define/boundary-conditions/set/<type>` command. Note that some settings will not take effect until after you have exited the command, by pressing **q**.

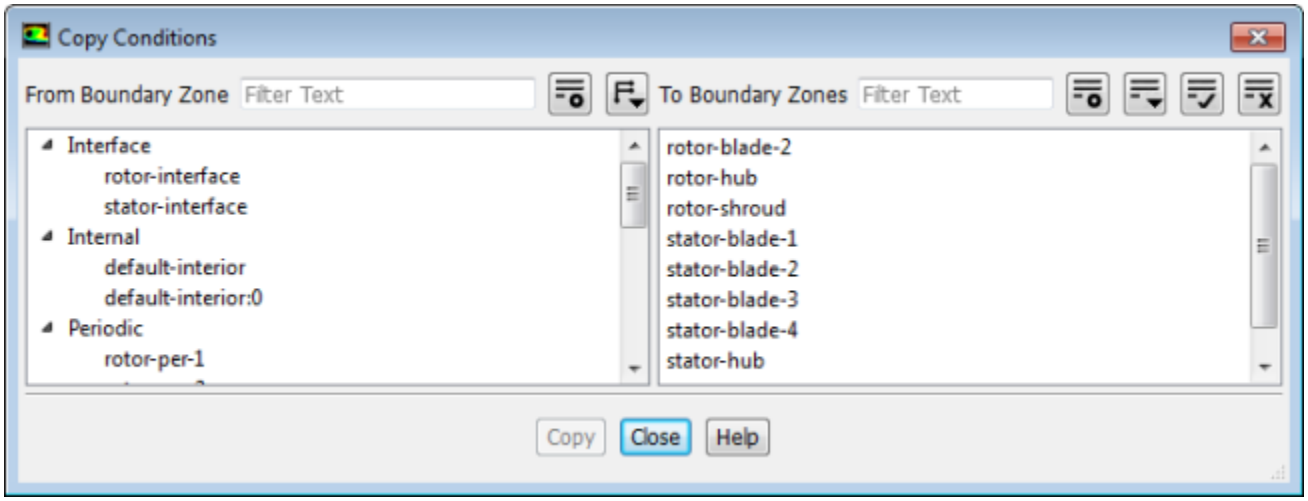
6.1.5. Copying Cell Zone and Boundary Conditions

You can copy cell zones and boundary conditions from one zone to other zones of the same type. If, for example, you have several wall zones in your model and they all have the same boundary conditions, you can set the conditions for one wall, and then simply copy them to the others.

The procedure for copying cell zone or boundary conditions is as follows:

1. Right-click the name of zone in the tree that you want to copy (under **Setup**/**Boundary Conditions** or **Setup**/**Cell Zone Conditions**) and select **Copy...**. This will open the **Copy Conditions** dialog box (Figure 6.2: The Copy Conditions Dialog Box (p. 242)).

   Figure 6.2: The Copy Conditions Dialog Box



2. In the **From Cell Zone** or **From Boundary Zone** list, confirm that the zone that has the conditions you want to copy is selected.

3. In the **To Cell Zones** or **To Boundary Zones** list, select the zone or zones to which you want to copy the conditions.

4. Click **Copy**. ANSYS Fluent will set *all* of the cell zones or boundary conditions for the zones selected in the **To Cell Zones** or **To Boundary Zones** list to be the same as the conditions for the zone selected in the **From Cell Zone** or **From Boundary Zone** list. (You cannot copy a subset of the conditions, such as only the thermal conditions.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that you cannot copy conditions from external walls to internal (that is, two-sided) walls, or vice versa, if the energy equation is being solved, since the thermal conditions for external and internal walls are different.

Important

If you are using one of the general multiphase models (VOF, mixture, or Eulerian), the procedure for copying boundary conditions is slightly different. See Steps for Copying Cell Zone and Boundary Conditions (p. 1406) for details.

6.1.6. Changing Cell or Boundary Zone Names

The default name for a zone is <*type*> **-** <*ID*> (for example, **pressure-inlet-7**). In some cases, you may want to assign more descriptive names to the boundary zones. If you have two pressure-inlet zones, for example, you might want to rename them **small-inlet** and **large-inlet**. (Changing the name of a zone will *not* change its type. Instructions for changing a zone's type are provided in Changing Cell and Boundary Zone Types (p. 239).)

To rename a zone, follow these steps:

1. Right-click the name of zone in the tree (under **Setup**/**Boundary Conditions** or **Setup**/**Cell Zone Conditions**).
2. Select **Edit...** to open the dialog box for the selected zone.
3. Enter a new name under **Zone Name**.
4. Click the **OK** button.

Note

If you specify a non-default name for a zone and then change its type, the name you specified will be retained; the automatic name change that accompanies a change in type occurs only for those with the default name.

6.1.7. Defining Non-Uniform Cell Zone and Boundary Conditions

Most conditions at each type of boundary zone can be defined as profile functions instead of constant values. You can use a profile contained in an externally generated profile file, or a function that you create using a user-defined function (UDF). Profiles are described in Profiles (p. 435), and user-defined functions are described in the separate Fluent Customization Manual.

6.1.8. Defining and Viewing Parameters

You can define a series of cases based on a set of parametric values. These parameters may be defined for numeric cell zone and boundary condition settings. This is especially useful if you are using Workbench and performing parametric studies (optimization), comparing cases with different boundary settings. Information about this feature can be found in see Working with Parameters and Design Points in the Workbench User's Guide. If you are not running ANSYS Fluent through Workbench, then you can use

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

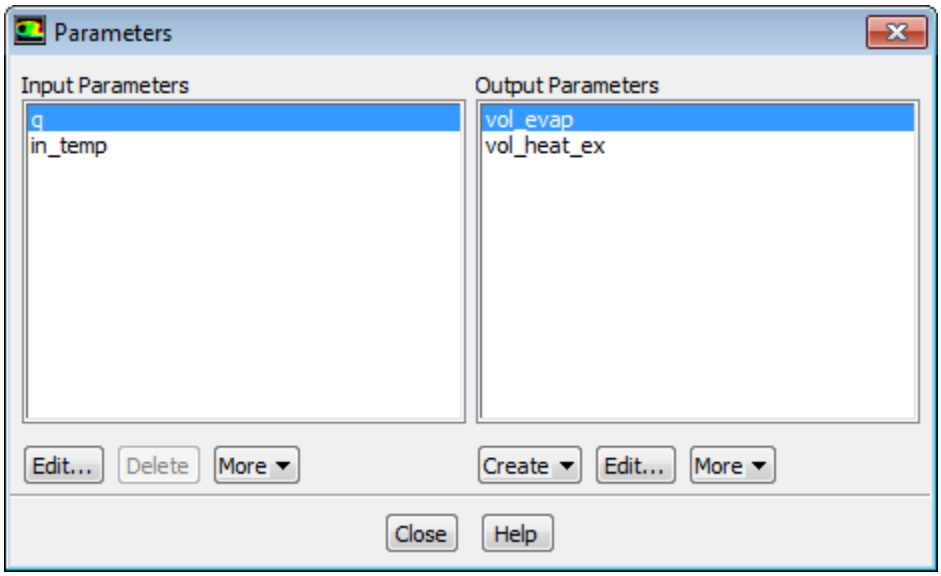
the parameter settings to define the same boundary condition value to different boundaries having the same units.

Note

For more information about using parameters with ANSYS Fluent in ANSYS Workbench, see Working With Input and Output Parameters in Workbench in the separate Fluent in Workbench User's Guide.

The parameters that you have defined in the various boundary condition dialog boxes are accessed by clicking the **Parameters...** button in the **Cell Zone Conditions** or **Boundary Conditions** task page. The **Parameters** dialog box will open, as shown in Figure 6.3: The Parameters Dialog Box (p. 244), listing all of the input parameters that you have created in the various boundary condition dialog boxes.

Figure 6.3: The Parameters Dialog Box



In the **Parameters** dialog box, under the **Input Parameters** list, you can

- Edit the input properties using the **Input Parameter Properties** dialog box, which is the same dialog box used to create parameters. This dialog box can also be accessed by selecting **New Input Parameter...** from the drop-down lists in the boundary conditions dialog boxes, as described later in this section.

  Important

  If you are using ANSYS Fluent in ANSYS Workbench, you cannot edit the input parameters, you can only view them. For more information, see the separate ANSYS Fluent in ANSYS Workbench User's Guide.

- Delete input parameters that are not assigned to a setting.
- More options exist under the **More** drop-down list (see Working With Advanced Parameter Options (p. 247)):

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use in Scheme Procedure
displays the Use Input Parameter in Scheme Procedure Dialog Box (p. 2628) where you can apply an input parameter using a Scheme procedure.

Use in UDF
displays the Use Input Parameter for UDF Dialog Box (p. 2629) where you can make an input parameter available in a user-defined function.

In the **Parameters** dialog box, under the **Output Parameters** list, you can also

- Create output parameters. These are single values generated by existing reports or monitors. You can generate the output parameters for **Fluxes**, **Forces**, **Surface**, **Volume**, **Drag**, **Lift**, **Moments**, and **User Defined** report definitions. These output parameters are discussed in greater detail in Creating Output Parameters (p. 1909).
- Edit existing output parameters.
- More options exist under the **More** drop-down list:

  Delete
  removes the selected output parameter from the list of **Output Parameters**.

  Rename
  allows you to edit the name of the output parameter through the **Rename** dialog box.

  Print to Console
  will output values to the console window. If you select multiple output parameters, then the output includes values from multiple output parameters.

  Print All to Console
  outputs the values from all output parameters to the console window.

  Write...
  allows you to store the output to a file. A dialog box is displayed allowing you to provide a file name.

  Write All...
  prompts you for a file name and then writes the values for all of the output parameters to a file.

Important

Changing the units for a quantity changes the value for any input parameter using that quantity.

Note

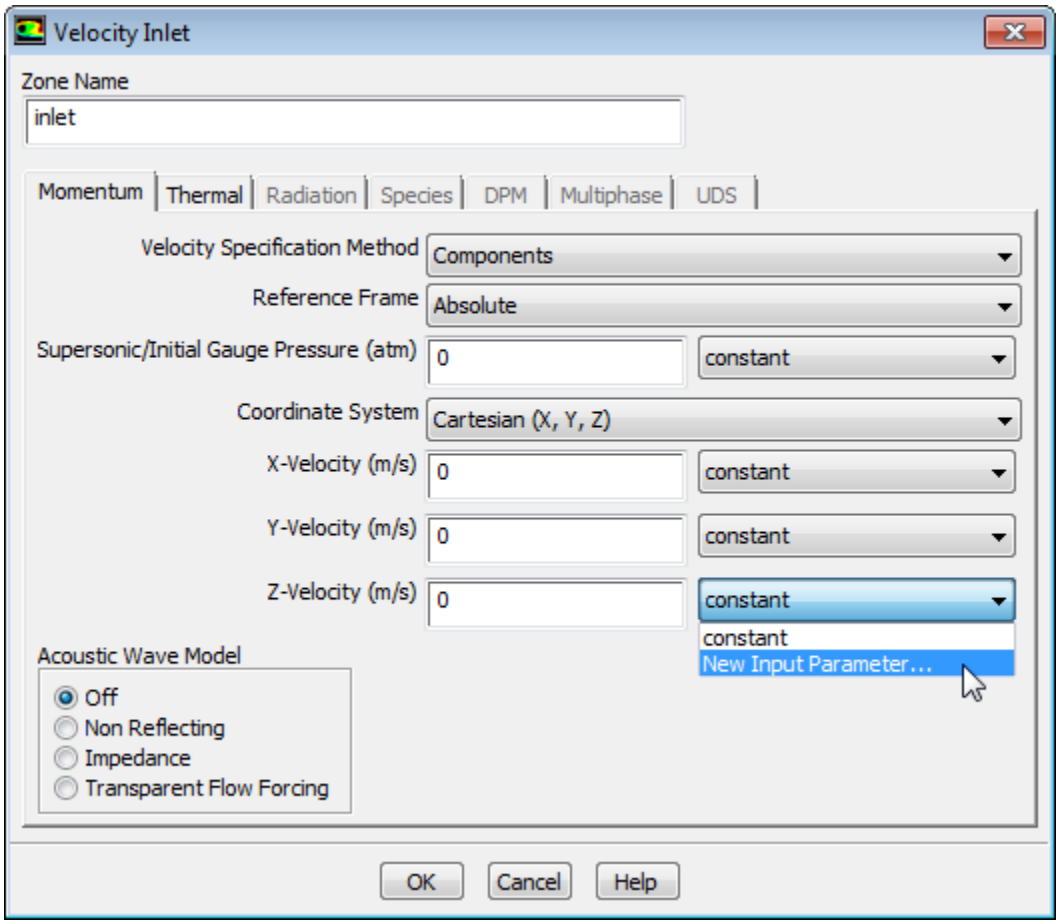
Various ANSYS Fluent setup-related input quantities (of type real and profile) can be assigned to an input parameter (indicated by the **New Input Parameter...** option in the corresponding drop-down list or by a small "p" icon next to the field). Clicking this option or icon displays the Select Input Parameter Dialog Box (p. 2384) where you can create and assign input parameters.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.1.8.1. Creating a New Parameter

You can create a new cell zone or boundary condition parameter, as shown in Figure 6.4: The New Input Parameter... Selection (p. 246).

Figure 6.4: The New Input Parameter... Selection

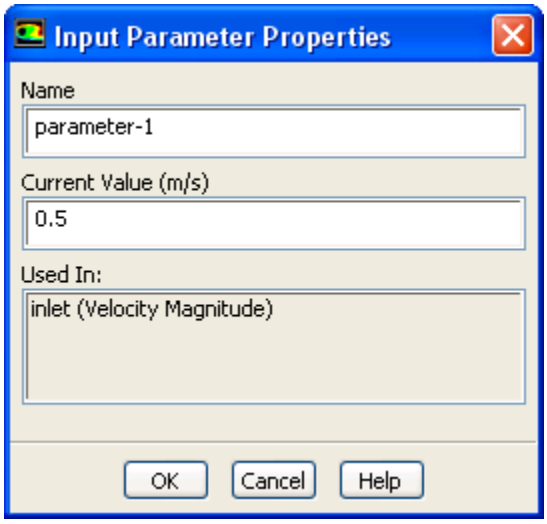


When you select **New Input Parameter...** from the drop-down list, the **Input Parameter Properties** dialog box (Figure 6.5: The Input Parameter Properties Dialog Box (p. 247)) will open where you will

- Enter the **Name** of the parameter.
- Specify the **Current Value** as a constant.
- The **Used In** field displays information about where the parameter is utilized.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.5: The Input Parameter Properties Dialog Box



Once the parameters are defined in the **Input Parameter Properties** dialog box, the name of the parameter will appear in the drop-down list of the property you are defining, as seen in Figure 6.4: The New Input Parameter... Selection (p. 246).

Note

ANSYS Fluent automatically creates generic default names for new input and output parameters (for example, `parameter-1`, `parameter-2`, `parameter-3`, and so on) If a parameter is deleted, the default name is not reused. For example, if you have `parameter-1`, `parameter-2`, and `parameter-3`, then delete `parameter-2` and create a new parameter, the default name for the new parameter will be `parameter-4`.

6.1.8.2. Working With Advanced Parameter Options

Various ANSYS Fluent setup related input quantities can be assigned to an input parameter. You can define a series of simulations based on a set of parametric values that are managed both in ANSYS Fluent and in Workbench. These parameters may be defined for numeric cell zone and boundary condition settings using the **New Input Parameter ...** option in the corresponding drop-down list or by a small "p" icon adjacent to a specific input setting. However, various ANSYS Fluent settings are not supported by these methods.

You can mitigate this limitation using input parameters through Scheme procedures and user-defined functions (UDFs), and define input parameters for various ANSYS Fluent simulation related settings.

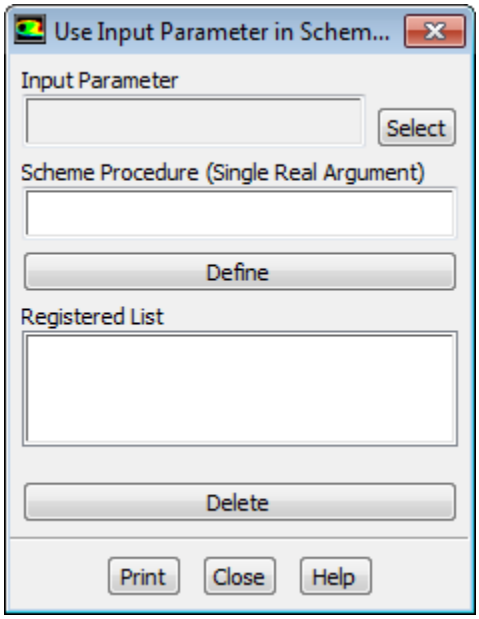
6.1.8.2.1. Defining Scheme Procedures With Input Parameters
6.1.8.2.2. Defining UDFs With Input Parameters
6.1.8.2.3. Using the Text User Interface to Define UDFs and Scheme Procedures With Input Parameters

6.1.8.2.1. Defining Scheme Procedures With Input Parameters

To use a Scheme procedure with input parameters, in the **Parameters** dialog box under **Input Parameters**, select **More**, then choose **Use in Scheme Procedure** to display the Use Input Parameter in Scheme Procedure Dialog Box (p. 2628) where you can register the input parameters that can be used in Scheme procedures during calculations. A text user interface command is also available, described in Using the Text User Interface to Define UDFs and Scheme Procedures With Input Parameters (p. 249).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.6: Use Input Parameter in Scheme Procedure Dialog Box



Use the **Select** button to open the Select Input Parameter Dialog Box (p. 2384) where an input parameter can be chosen. Enter a **Scheme Procedure** name or body starting with `lambda` that should receive a real argument (the value of the selected input parameter). For example, if the following Scheme procedure called `my-funct` is defined as a Scheme file that gets loaded into the current session,

```
(define my-funct
 (lambda (value )
  (ti-menu-load-string (format #f "/solve/set/under-relaxation/pressure ~g" value))))
```

then enter `my-funct` for the **Scheme Procedure**. Otherwise, you can directly enter the following code for the **Scheme Procedure**:

```
(lambda (value)(ti-menu-load-string (args->string '/solve/set/under-relaxation/pressure value)))
```

Click **Define** to populate the **Registered List**. Use **Delete** to delete the use of the selected input variable, but not the associated input parameter (the input parameter has to be deleted separately in the **Parameters** dialog box). Use **Print** to print out details of the registered input parameters.

Important

While writing Scheme procedures, you should use the "args->string" help procedure instead of using double quotes. For example:

```
(lambda ( value1)
 (display (args->string 'Changing-Theta-Divisions-value=   value1))
 (ti-menu-load-string
    (args->string '/define/models/radiation/discrete-ordinates? 'yes value1  '_  '_  '_)) )
```

Alternatively, you could also write:

```
(lambda ( value1)
 (display (args->string 'Changing-Theta-Divisions-value=  value1))
 (ti-menu-load-string
   (apply args->string `(/define/models/radiation/discrete-ordinates? yes ,value1  _  _  _) )) )
```

where the ` character after `args->string` is the grave accent character (not a single quote) and the _ character stores the default value of the current text user interface prompt. Note

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

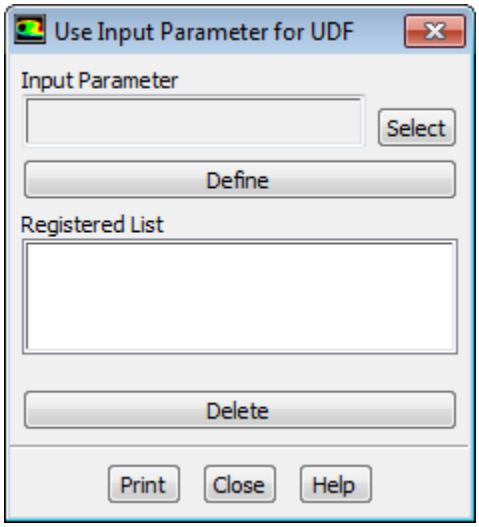
that the first occurrence of `value1` is the Scheme variable, while the second occurrence of `value1` will be replaced by the value stored in the variable. This method eliminates the need to specify a single quote for all arguments.

Also note that if you enter a scheme procedure body in the **Use Input Parameter in Scheme Procedure** dialog box, you should combine the scheme procedure lines into one single line.

6.1.8.2.2. Defining UDFs With Input Parameters

To use a UDF with input parameters, in the **Parameters** dialog box under **Input Parameters**, select **More**, then choose **Use in UDF** to display the Use Input Parameter for UDF Dialog Box (p. 2629) where you can register the input parameters that can be used in UDF functions during calculations. A text user interface command is also available, described in Using the Text User Interface to Define UDFs and Scheme Procedures With Input Parameters (p. 249).

Figure 6.7: Use Input Parameter for UDF Dialog Box



Use the **Select** button to open the Select Input Parameter Dialog Box (p. 2384) where an input parameter can be chosen. Click **Define** to populate the **Registered List**. Use **Delete** to delete the use of the selected input variable, but not the associated input parameter (the input parameter has to be deleted separately in the **Parameters** dialog box). Click **Print** to print out the parameter ID of the registered input parameter. This ID should be used to access the value of the parameter in UDF.

By registering the input parameter, ANSYS Fluent knows that this input parameter is being used by a UDF. In the UDF C function, you can access the registered input parameter value using the `RP_Get_Input_Parameter` macro. For example,

```
real value = RP_Get_Input_Parameter(“real-4”),
```

where `real-4` is an input parameter ID obtained using the **Print** feature.

6.1.8.2.3. Using the Text User Interface to Define UDFs and Scheme Procedures With Input Parameters

The `/define/parameters/input-parameters/advance/use-in` text interface command can also be used to define input parameters and Scheme procedures and user-defined functions.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Scheme procedures can use other text user interface (TUI) commands in ANSYS Fluent to change the desired simulation settings in a parametric manner. Each numerical component of the TUI command string can be marked and treated as a parameter. Setting up advanced input parameters requires using Scheme functions that use text interface commands to apply new values. Once defined, these input parameters are displayed in the **Parameters** dialog box alongside other input parameters. The input parameter passes a constant numeric value to the registered/provided Scheme function. Therefore, the associated Scheme function (and corresponding ANSYS Fluent text command) uses the constant parameter values using the units that were already defined for the designated text command quantity.

`/define/parameters/input-parameters/advance`

Enter the advanced input parameters menu.

`use-in/`

Allows you to use an input parameter in a Scheme procedure or in a UDF. The following examples demonstrates how to use an input parameter using a Scheme procedure. The Scheme procedure body can be written at the prompt itself:

```
/define/parameters/input-parameters/advance> use-in
Select type:> scheme-proc
Name of parameter ["parameter-1"]
parameter-1 value [0] 0.3
Enter the name/body of apply-function [()] (lambda (value )
        (ti-menu-load-string (format #f "/solve/set/under-relaxation/pressure ~g" value)))
```

or it can be written in a Scheme file (for example, `my-funct`)

```
/define/parameters/custom-input-parameters> use-in
Select type:> scheme-proc
Name of parameter ["parameter-1"]
parameter-1 value [0] 0.3
Enter the name/body of apply-function [()] my-funct
```

where the `my-funct` Scheme file contents are:

```
(define my-funct
    (lambda (value )
        (ti-menu-load-string (format #f "/solve/set/under-relaxation/pressure ~g" value))))
```

The following example demonstrates how to use an input parameter using a UDF:

```
/define/parameters/input-parameters/advance> use-in
Select type:> udf-side
Name of parameter ["parameter-1"]
parameter-1 value [0] 0.3
```

`delete`

Deletes the use of the selected input variable, but not the associated input parameter (the input parameter has to be deleted separately). Using the wildcard '*' allows you to delete all custom input variable at once.

`list`

Shows usage of input parameters for Scheme procedures and for UDFs

6.1.9. Selecting Cell or Boundary Zones in the Graphics Display

Whenever you need to select a zone in the **Cell Zone Conditions** or **Boundary Conditions** task page, you can use the mouse in the graphics window to choose the appropriate zone. This feature is particularly useful if you are setting up a problem for the first time, or if you have two or more zones of the same type and you want to determine the zone IDs (that is, figure out which zone is which). To use this feature, do the following:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Display the mesh using the Mesh Display Dialog Box (p. 2163).

2. Use the mouse probe button (the right button, by default—see Controlling the Mouse Button Functions (p. 1800) to modify the mouse button functions) to click a cell or boundary zone in the graphics window.

The zone you select in the graphics display will automatically be selected in the **Zone** list in the **Cell Zone Conditions** or **Boundary Conditions** task page, and its name and ID will be printed in the console window.

However, if you prefer to select the surfaces to display using the **Mesh Display** dialog box, then simply click the **Display Mesh...** button in the **Cell Zone Conditions** or **Boundary Conditions** task page.

Setup → **Cell Zone Conditions** → **Display Mesh...**

Setup → **Boundary Conditions** → **Display Mesh...**

Detailed information about the **Mesh Display** dialog box can be found in Displaying the Mesh (p. 1748).

6.1.10. Operating and Periodic Conditions

The **Cell Zone Conditions** and **Boundary Conditions** task pages allow you to access the **Operating Conditions** dialog box, where you can set the operating pressure, reference pressure location, include the effects of gravity, and specify other operating variables, as discussed in Modeling Basic Fluid Flow (p. 573).

Setup → **Cell Zone Conditions** → **Operating Conditions...**

Setup → **Boundary Conditions** → **Operating Conditions...**

The **Periodic Conditions** dialog box can be accessed from the **Boundary Conditions** task page. For a detailed description of this dialog box's inputs, refer to Periodic Flows (p. 582).

Setup → **Boundary Conditions** → **Periodic Conditions...**

6.1.11. Highlighting Selected Boundary Zones

To highlight a selected boundary zone of a model that is displayed in the graphics window, enable the **Highlight Zone** option in the **Boundary Conditions** task page.

Important

Note that this is only applicable to 3D cases and is available only in the **Boundary Conditions** task page.

There are two ways in which you can highlight a boundary zone in the graphics window, after enabling the **Highlight Zone** option:

- Select the zone by highlighting it in the **Zone** list, in the **Boundary Conditions** task page.
- Use the mouse-probe button and select the zone in the graphics window.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The selected zone in the graphics window will be highlighted in a cyan color, which is the default color. You can change the color using the following text command:

`display → set → colors → highlight-color`

If you want to highlight a boundary zone that is not displayed in the graphics window, ANSYS Fluent will display that boundary zone in the graphics window and then highlight it. If you disable the **Highlight Zone** option, the displayed boundary zone will be removed from the scene (graphics window) and your model will be redrawn to its original view.

6.1.12. Saving and Reusing Cell Zone and Boundary Conditions

You can save cell zone and boundary conditions to a file so that you can use them to specify the same conditions for a different case, as described in Reading and Writing Boundary Conditions (p. 56).

6.2. Cell Zone Conditions

Cell zones consist of fluids and solids. Porous zones and 3D fans in ANSYS Fluent are treated as fluid zones. A detailed description of the various cell zones is given in the sections that follow.

6.2.1. Fluid Conditions
6.2.2. Solid Conditions
6.2.3. Porous Media Conditions
6.2.4. 3D Fan Zones
6.2.5. Fixing the Values of Variables
6.2.6. Locking the Temperature for Solid and Shell Zones
6.2.7. Defining Mass, Momentum, Energy, and Other Sources

6.2.1. Fluid Conditions

A fluid zone is a group of cells for which all active equations are solved. The only required input for a fluid zone is the type of fluid material. You must indicate which material the fluid zone contains so that the appropriate material properties will be used.

Important

If you are modeling multiphase flow, you will not specify the materials here; you will choose the phase material when you define the phases, as described in Defining the Phases for the VOF Model (p. 1436).

Important

If you are modeling species transport and/or combustion, you can specify the material as either a mixture or a fluid. The mixture material has to be the same as that specified in the **Species Model** dialog box when you enable the model. The fluid zones, being of different material types, must not be contiguous.

Optional inputs allow you to set sources or fixed values of mass, momentum, heat (temperature), turbulence, species, and other scalar quantities. You can also define motion for the fluid zone. If there are rotationally periodic boundaries adjacent to the fluid zone, you must specify the rotation axis. If you are modeling turbulence using one of the k- ε models, the k- ω model, or the Spalart-Allmaras model,

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

you can choose to define the fluid zone as a laminar flow region. If you are modeling radiation using the DO model, you can specify whether or not the fluid participates in radiation.

Warning

In general, disabling **Participates in Radiation** for fluid zones is not recommended, as it can produce erroneous results. There are rare cases when it is acceptable: for example, if the domain contains multiple fluid zones, disabling this option for zones where radiation is negligible may save computational time without affecting the results.

Important

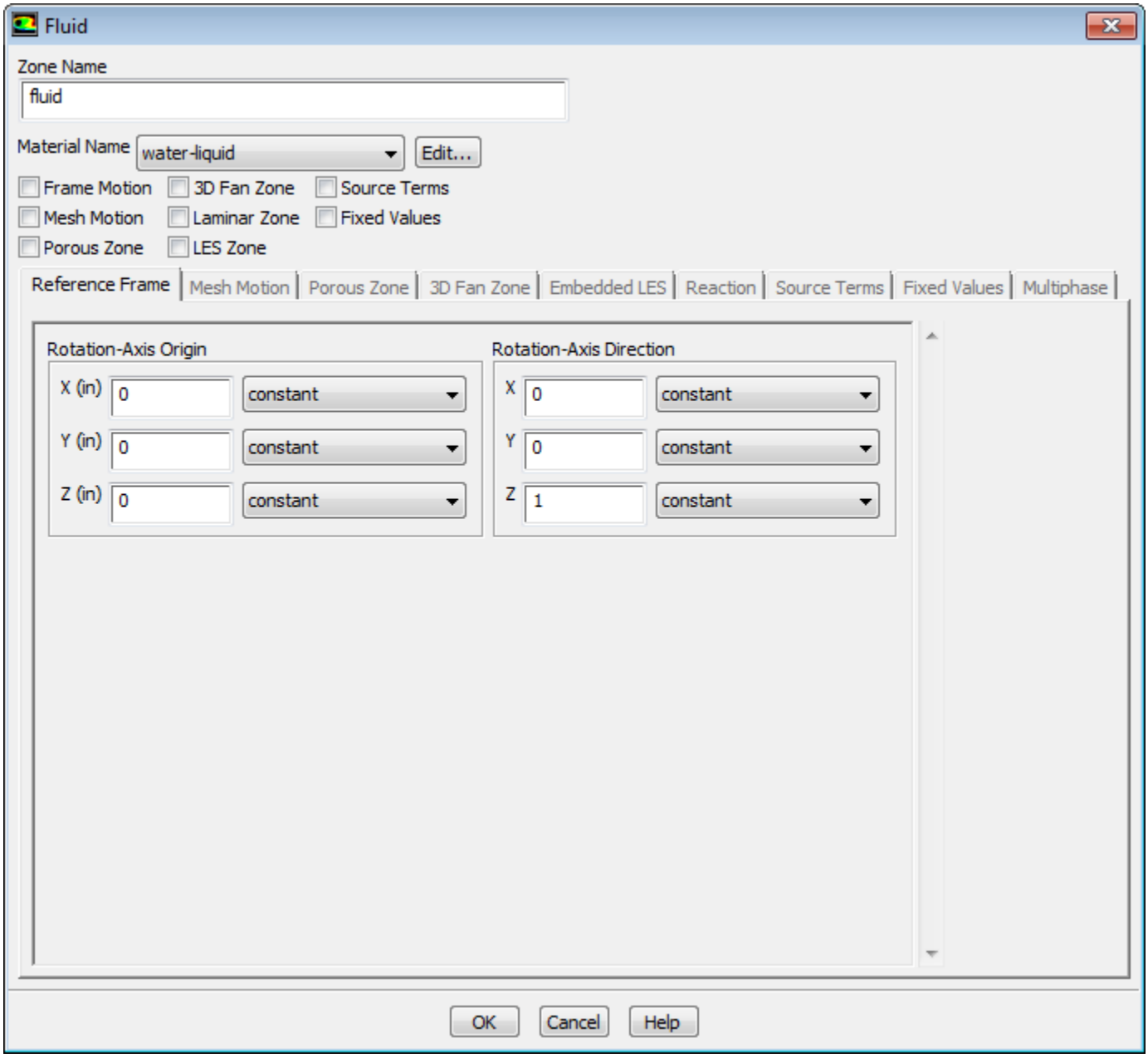
For information about porous zones and 3D fan zones, see Porous Media Conditions (p. 261) and 3D Fan Zones (p. 287), respectively.

6.2.1.1. Inputs for Fluid Zones

You will set all fluid conditions in the Fluid Dialog Box (p. 2369) (Figure 6.8: The Fluid Dialog Box (p. 254)), which is accessed from the **Cell Zone Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 241)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.8: The Fluid Dialog Box



6.2.1.1.1. Defining the Fluid Material

To define the material contained in the fluid zone, select the appropriate item in the **Material Name** drop-down list. This list will contain all fluid materials database) in the Create/Edit Materials Dialog Box (p. 2302). If you want to check or modify the properties of the selected material, you can click **Edit...** to open the **Edit Material** dialog box; this dialog box contains just the properties of the selected material, not the full contents of the standard **Create/Edit Materials** dialog box.

Important

If you are modeling species transport or multiphase flow, the **Material Name** list will not appear in the **Fluid** dialog box. For species calculations, the mixture material for all fluid zones will be the material you specified in the Species Model Dialog Box (p. 2216). For mul-

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tiphase flows, the materials are specified when you define the phases, as described in Defining the Phases for the VOF Model (p. 1436).

6.2.1.1.2. Defining Sources

If you want to define a source of heat, mass, momentum, turbulence, species, or other scalar quantity within the fluid zone, you can do so by enabling the **Source Terms** option. See Defining Mass, Momentum, Energy, and Other Sources (p. 297) for details.

6.2.1.1.3. Defining Fixed Values

If you want to fix the value of one or more variables in the fluid zone, rather than computing them during the calculation, you can do so by enabling the **Fixed Values** option. See Fixing the Values of Variables (p. 292) for details.

6.2.1.1.4. Specifying a Laminar Zone

When you are calculating a turbulent flow, it is possible to "turn off" turbulence modeling in a specific fluid zone. To disable turbulence modeling, turn on the **Laminar Zone** option in the **Fluid** dialog box. This is useful if you know that the flow in a certain region is laminar. For example, if you know the location of the transition point on an airfoil, you can create a laminar/turbulent transition boundary where the laminar cell zone borders the turbulent cell zone. This feature allows you to model turbulent transition on the airfoil.

By default, the **Laminar Zone** option will set the turbulent viscosity, μ_t, to zero and disable turbulence production in the fluid zone. Turbulent quantities will still be transported through the zone, but effects on fluid mixing and momentum will be ignored. If you want to keep the turbulent viscosity, you can do so using the text command `define/ boundary-conditions/fluid`. You will be asked if you want to `Set Turbulent Viscosity to zero within laminar zone?`. If your response is `no`, ANSYS Fluent will set the production term in the turbulence transport equation to zero, but will retain a nonzero μ_t.

Disabling turbulence modeling in a fluid zone can be applied to all the turbulence models except the Large Eddy Simulation (LES) model.

6.2.1.1.5. Specifying a Reaction Mechanism

If you are modeling species transport with reactions, you can enable a reaction mechanism in a fluid zone by turning on the **Reaction** option and selecting an available mechanism from the **Reaction Mechanism** drop-down list. See Defining Zone-Based Reaction Mechanisms (p. 984) for more information about defining reaction mechanisms.

6.2.1.1.6. Specifying the Rotation Axis

If there are rotationally periodic boundaries adjacent to the fluid zone or if the zone is rotating, either the mesh or its reference frame, you must specify the rotation axis. To define the axis for a moving reference frame problem, set the **Rotation-Axis Direction** and **Rotation-Axis Origin** under the **Reference**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Frame tab. To define the axis for a moving mesh problem, set the **Rotation-Axis Direction** and **Rotation-Axis Origin** under the **Mesh Motion** tab.

Note

If a frame motion and a mesh motion are specified at the same zone and this zone has rotationally periodic boundaries adjacent to it, then both axes have to be coaxial. Otherwise, the periodicity assumption is not valid and you will receive a warning message. In addition, the mesh check will fail.

The cell zone axis is independent of the axis of rotation used by any adjacent wall zones or any other cell zones. For 3D problems, the axis of rotation is the vector from the **Rotation-Axis Origin** in the direction of the vector given by your **Rotation-Axis Direction** inputs for the frame of reference and the mesh motion. For 2D non-axisymmetric problems, you will specify only the **Rotation-Axis Origin**; the axis of rotation is the z-direction vector passing through the specified point. (The z direction is normal to the plane of your geometry so that rotation occurs in the plane.) For 2D axisymmetric problems, you will not define the axis: the rotation will always be about the x axis, with the origin at (0,0).

6.2.1.1.7. Defining Zone Motion

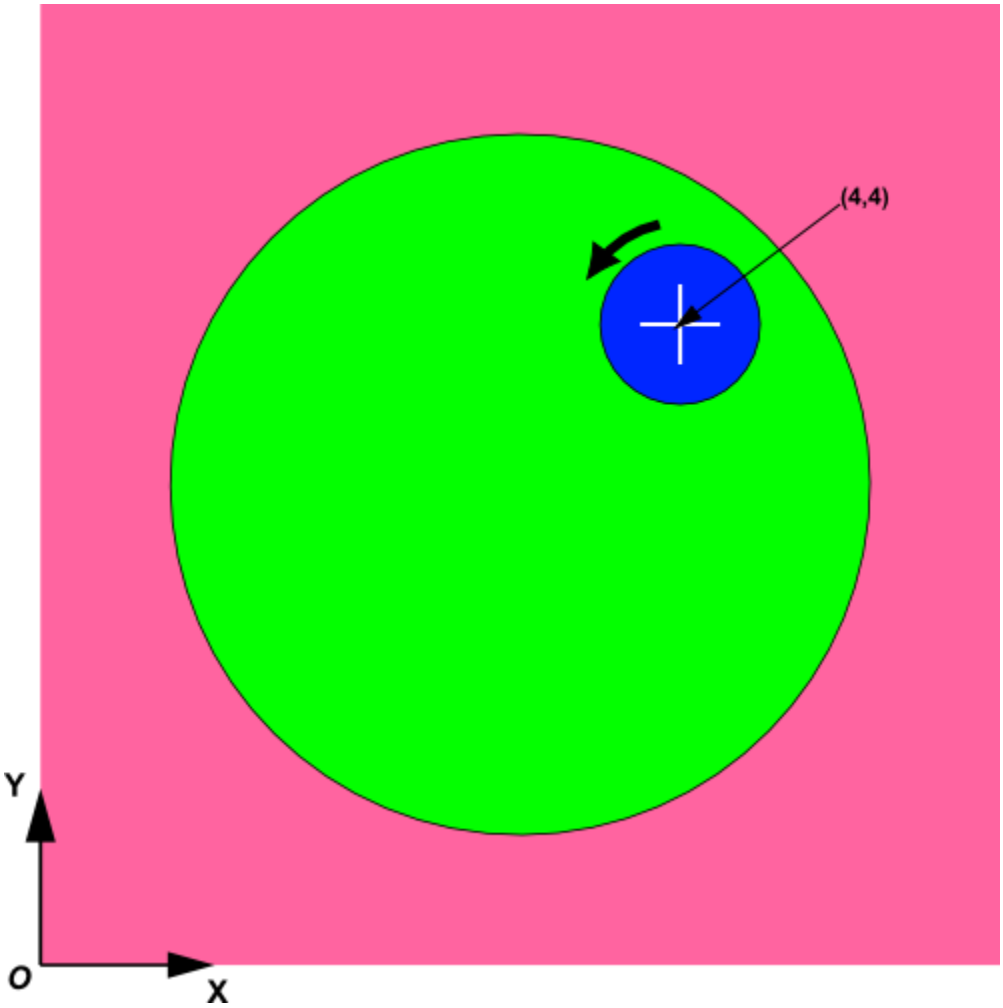
To define zone motion for a moving reference frame (MRF), enable the **Frame Motion** option in the **Fluid** dialog box. Set the appropriate parameters in the expanded portion of the dialog box, under the **Reference Frame** tab.

To define zone motion for a moving (sliding) mesh, enable the **Mesh Motion** option in the **Fluid** dialog box. Set the appropriate parameters in the expanded portion of the dialog box, under the **Mesh Motion** tab. See Setting Up the Sliding Mesh Problem (p. 634) for details.

For cases that do not contain zones with motion specified in a relative frame to another zone, select **absolute** from the **Relative To Cell Zone** drop-down list. Here, the velocity and rotation components are specified in an **absolute** reference frame, which is the default setting, as shown in Figure 6.9: Rotation Specified in the Absolute Reference Frame (p. 257). If no moving zones are present in the simulation, then **absolute** will be the only available selection. See The Multiple Reference Frame Model for more information.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.9: Rotation Specified in the Absolute Reference Frame

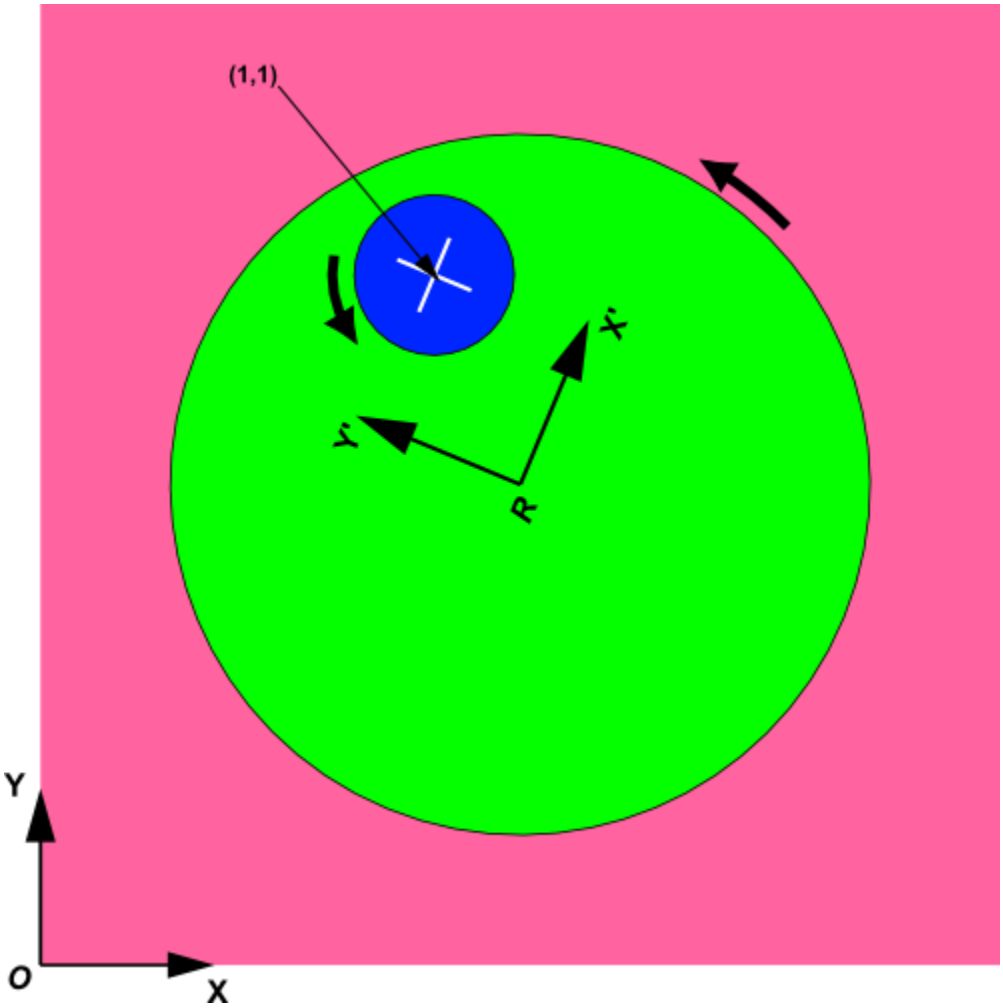


For cases where you have a moving zone specified relative to another moving zone, select the cell zone carrying the primary motion from the **Relative To Cell Zone** drop-down list under the **Reference Frame** tab or the **Mesh Motion** tab. Note that for such cases, **Rotation-Axis Origin (Relative)** will appear in the interface, signifying coordinates relative to the zone selected from the **Relative To Cell Zone** drop-down list.

Figure 6.10: Rotation Specified Relative to a Moving Zone (p. 258) illustrates that the rotational axis origin of the small rotating zone is specified relative to the cell zone carrying the primary motion (having local coordinate system R).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.10: Rotation Specified Relative to a Moving Zone



Note

The **Relative To Cell Zone** list will consist of all moving cell zones with an absolute motion specification (that is zones that are moving, but their motion is not relative to some other zone), excluding the current cell zone.

For problems that include linear, translational motion of the fluid zone, specify the **Translational Velocity** by setting the **X**, **Y**, and **Z** components under the **Mesh Motion** tab. For problems that include rotational motion, specify the rotational **Speed** under **Rotational Velocity**. The rotation axis is defined as described above. Note that the speed can be specified as a constant value or a transient profile. The transient profile may be in a file format, as described in Defining Transient Cell Zone and Boundary Conditions (p. 448), or a UDF macro (DEFINE_TRANSIENT_PROFILE). Specifying the individual velocities as either a profile or a UDF allows you to specify a single component of the frame motion individually. However, you can also specify the frame motion using a user-defined function. This may prove to be quite con-

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

venient if you are modeling a more complicated motion of the moving reference frame, where the hooking of many different user-defined functions or profiles can be cumbersome.

Important

If you need to switch between the MRF and moving mesh models, simply click the **Copy To Mesh Motion** for zones with a moving frame of reference and **Copy to Frame Motion** for zones with moving meshes to transfer motion variables, such as the axes, frame origin, and velocity components between the two models. The variables used for the origin, axis, and velocity components, as well as for the UDF `DEFINE_ZONE_MOTION` will be copied. This is particularly useful if you are doing a steady-state MRF simulation to obtain an initial solution for a transient Moving Mesh simulation in a turbomachine.

See Modeling Flows with Moving Reference Frames (p. 603) for details about modeling flows in moving reference frames. Details about the frame motion UDF can be found in `DEFINE_ZONE_MOTION` in the Fluent Customization Manual.

6.2.1.1.8. Defining Radiation Parameters

If you are using the DO radiation model, you can specify whether or not the fluid zone participates in radiation using the **Participates in Radiation** option. See Defining Boundary Conditions for Radiation (p. 869) for details.

6.2.2. Solid Conditions

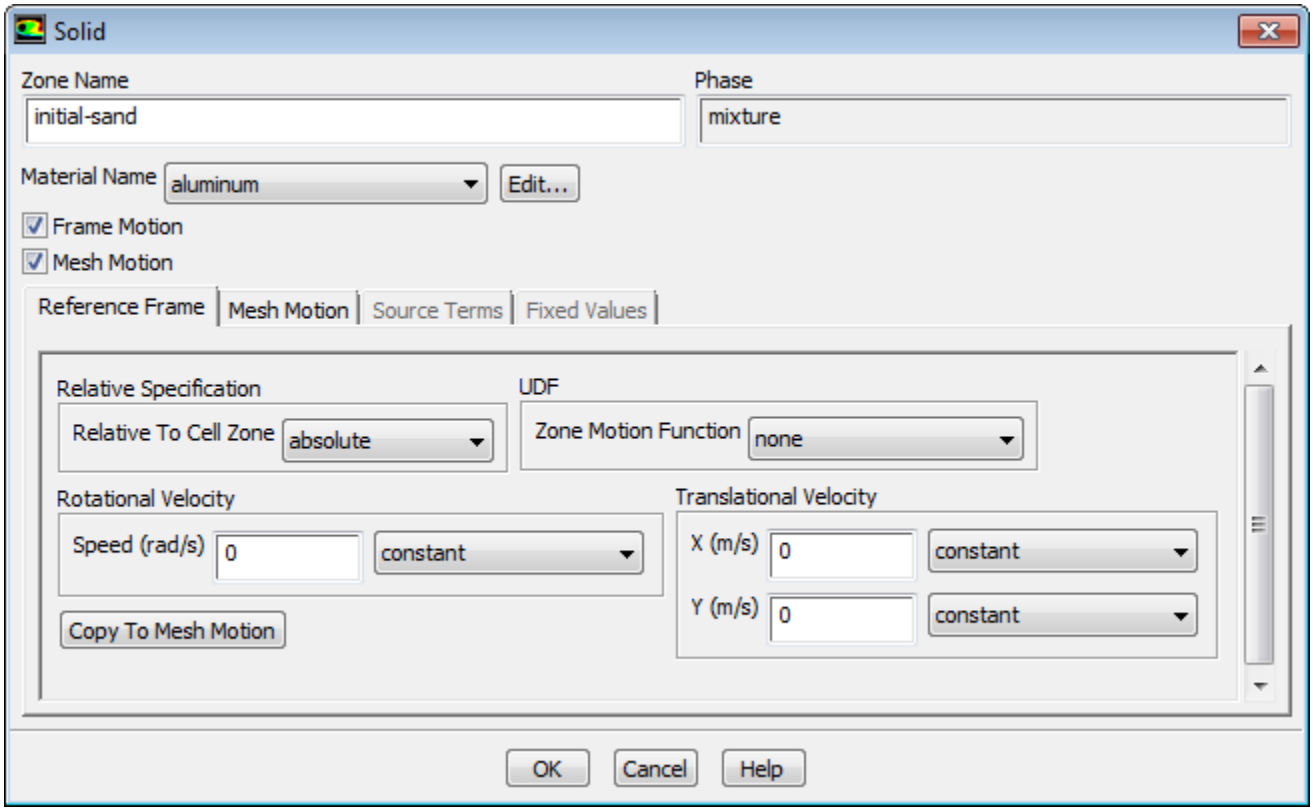
A "solid" zone is a group of cells for which only a heat conduction problem is solved; no flow equations are solved. The material being treated as a solid may actually be a fluid, but it is assumed that no convection is taking place. The only required input for a solid zone is the type of solid material. You must indicate which material the solid zone contains so that the appropriate material properties will be used. Optional inputs allow you to set a volumetric heat generation rate (heat source) or a fixed value of temperature. You can also define motion for the solid zone. If there are rotationally periodic boundaries adjacent to the solid zone, you must specify the rotation axis. If you are modeling radiation using the DO model, you can specify whether or not the solid material participates in radiation.

6.2.2.1. Inputs for Solid Zones

You will set all solid conditions in the Solid Dialog Box (p. 2379) (Figure 6.11: The Solid Dialog Box (p. 260)), which is opened from the **Cell Zone Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 241)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.11: The Solid Dialog Box



6.2.2.1.1. Defining the Solid Material

To define the material contained in the solid zone, select the appropriate item in the **Material Name** drop-down list. This list will contain all solid materials database) in the Create/Edit Materials Dialog Box (p. 2302). If you want to check or modify the properties of the selected material, you can click **Edit...** to open the **Edit Material** dialog box; this dialog box contains just the properties of the selected material, not the full contents of the standard **Create/Edit Materials** dialog box.

6.2.2.1.2. Defining a Heat Source

If you want to define a source of heat within the solid zone, you can do so by enabling the **Source Terms** option. See Defining Mass, Momentum, Energy, and Other Sources (p. 297) for details.

6.2.2.1.3. Defining a Fixed Temperature

If you want to fix the value of temperature in the solid zone, rather than computing it during the calculation, you can do so by enabling the **Fixed Values** option. See Fixing the Values of Variables (p. 292) for details.

6.2.2.1.4. Specifying the Rotation Axis

If there are rotationally periodic boundaries adjacent to the solid zone or if the zone is rotating, either the mesh or its reference frame, you must specify the rotation axis. To define the axis for a moving reference frame problem, set the **Rotation-Axis Direction** and **Rotation-Axis Origin** under the **Reference**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Frame tab. To define the axis for a moving mesh problem, set the **Rotation-Axis Direction** and **Rotation-Axis Origin** under the **Mesh Motion** tab.

Note

If a frame motion and a mesh motion are specified at the same zone and this zone has rotationally periodic boundaries adjacent to it, then both axes have to be coaxial. Otherwise, the periodicity assumption is not valid and you will receive a warning message. In addition, the mesh check will fail.

The cell zone axis is independent of the axis of rotation used by any adjacent wall zones or any other cell zones. For 3D problems, the axis of rotation is the vector from the **Rotation-Axis Origin** in the direction of the vector given by your **Rotation-Axis Direction** inputs for the frame of reference and the mesh motion. For 2D non-axisymmetric problems, you will specify only the **Rotation-Axis Origin**; the axis of rotation is the z-direction vector passing through the specified point. (The z direction is normal to the plane of your geometry so that rotation occurs in the plane.) For 2D axisymmetric problems, you will not define the axis: the rotation will always be about the x axis, with the origin at (0,0).

6.2.2.1.5. Defining Zone Motion

Defining zone motion in solids is similar to defining them for fluids. Refer to Defining Zone Motion (p. 256) for details.

6.2.2.1.6. Defining Radiation Parameters

If you are using the DO radiation model, you can specify whether or not the solid material participates in radiation using the **Participates in Radiation** option. See Defining Boundary Conditions for Radiation (p. 869) for details.

6.2.3. Porous Media Conditions

The porous media model can be used for a wide variety of single phase and multiphase problems, including flow through packed beds, filter papers, perforated plates, flow distributors, and tube banks. When you use this model, you define a cell zone in which the porous media model is applied and the pressure loss in the flow is determined via your inputs as described in Momentum Equations for Porous Media (p. 262). Heat transfer through the medium can also be represented, with or without the assumption of thermal equilibrium between the medium and the fluid flow (as described in Treatment of the Energy Equation in Porous Media (p. 265)).

A 1D simplification of the porous media model, termed the "porous jump," can be used to model a thin membrane with known velocity/pressure-drop characteristics. The porous jump model is applied to a face zone, not to a cell zone, and should be used (instead of the full porous media model) whenever possible because it is more robust and yields better convergence. See Porous Jump Boundary Conditions (p. 403) for details.

6.2.3.1. Limitations and Assumptions of the Porous Media Model

The porous media model incorporates an empirically determined flow resistance in a region of your model defined as "porous". In essence, the porous media model adds a momentum sink in the governing momentum equations. Consequently, the following modeling assumptions and limitations should be readily recognized:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Since the volume blockage that is physically present is not represented in the model, by default ANSYS Fluent uses and reports a superficial velocity inside the porous medium, based on the volumetric flow rate, to ensure continuity of the velocity vectors across the porous medium interface. This superficial velocity formulation does not take porosity into account when calculating the convection and diffusion terms of the transport equations. You can choose to use a more accurate alternative, in which the true (physical) velocity is calculated inside the porous medium and porosity is included in the differential terms of the transport equations. See Modeling Porous Media Based on Physical Velocity (p. 283) for details. In a multiphase flow system, all phases share the same porosity.

- The effect of the porous medium on the turbulence field is only approximated. See Treatment of Turbulence in Porous Media (p. 266) for details.

- In general, the ANSYS Fluent porous medium model, for both single phase and multiphase, assumes the porosity is isotropic, and it can vary with space and time.

- The Superficial Velocity Formulation and the Physical Velocity Formulation are available for multiphase porous media. See User Inputs for Porous Media (p. 267) for details.

- The porous media momentum resistance and heat source terms are calculated separately on each phase. See Momentum Equations for Porous Media (p. 262) for details.

- A unique pressure interpolation scheme is always used inside porous media zones regardless of the pressure scheme selected in the **Solution Methods** task page.

- The interactions between a porous medium and shock waves are not considered.

- When applying the porous media model in a moving reference frame, ANSYS Fluent will either apply the relative reference frame or the absolute reference frame when you enable the **Relative Velocity Resistance Formulation**. This allows for the correct prediction of the source terms.

- **Standard Initialization** is the recommended initialization method for porous media simulations. The default **Hybrid Initialization** method does not account for the porous media properties, and depending on boundary conditions, may produce an unrealistic initial velocity field. For porous media simulations, the **Hybrid Initialization** method can only be used if the **Maintain Constant Velocity Magnitude** option is selected in the **Hybrid Initialization** dialog box.

6.2.3.2. Momentum Equations for Porous Media

The porous media models for single phase flows and multiphase flows use the **Superficial Velocity Porous Formulation** as the default. ANSYS Fluent calculates the superficial phase or mixture velocities based on the volumetric flow rate in a porous region. The porous media model is described in the following sections for single phase flow, however, it is important to note the following for multiphase flow:

- In the Eulerian multiphase model (Eulerian Model Theory in the Theory Guide), the general porous media modeling approach, physical laws, and equations described below are applied to the corresponding phase for mass continuity, momentum, energy, and all the other scalar equations.

- The **Superficial Velocity Porous Formulation** generally gives good representations of the bulk pressure loss through a porous region. However, since the superficial velocity values within a porous region remain the same as those outside the porous region, it cannot predict the velocity increase in porous zones and therefore limits the accuracy of the model.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Porous media are modeled by the addition of a momentum source term to the standard fluid flow equations. The source term is composed of two parts: a viscous loss term (Darcy, the first term on the right-hand side of Equation 6.1 (p. 263), and an inertial loss term (the second term on the right-hand side of Equation 6.1 (p. 263))

$$S_i = -\left(\sum_{j=1}^{3} D_{ij} \mu v_j + \sum_{j=1}^{3} C_{ij} \frac{1}{2} \rho |v| v_j \right) \tag{6.1}$$

where S_i is the source term for the i th (x, y, or z) momentum equation, $|v|$ is the magnitude of the velocity and D and C are prescribed matrices. This momentum sink contributes to the pressure gradient in the porous cell, creating a pressure drop that is proportional to the fluid velocity (or velocity squared) in the cell.

To recover the case of simple homogeneous porous media

$$S_i = -\left(\frac{\mu}{\alpha} v_i + C_2 \frac{1}{2} \rho |v| v_i \right) \tag{6.2}$$

where α is the permeability and C_2 is the inertial resistance factor, simply specify D and C as diagonal matrices with $1/\alpha$ and C_2, respectively, on the diagonals (and zero for the other elements).

ANSYS Fluent also allows the source term to be modeled as a power law of the velocity magnitude:

$$S_i = -C_0 |v|^{C_1} = -C_0 |v|^{(C_1 - 1)} v_i \tag{6.3}$$

where C_0 and C_1 are user-defined empirical coefficients.

Important

In the power-law model, the pressure drop is isotropic and the units for C_0 are SI.

6.2.3.2.1. Darcy's Law in Porous Media

In laminar flows through porous media, the pressure drop is typically proportional to velocity and the constant C_2 can be considered to be zero. Ignoring convective acceleration and diffusion, the porous media model then reduces to Darcy's Law:

$$\nabla p = -\frac{\mu}{\alpha} \vec{v} \tag{6.4}$$

The pressure drop that ANSYS Fluent computes in each of the three (x, y, z) coordinate directions within the porous region is then

$$\begin{aligned} \Delta p_x &= \sum_{j=1}^{3} \frac{\mu}{\alpha_{xj}} v_j \Delta n_x \\ \Delta p_y &= \sum_{j=1}^{3} \frac{\mu}{\alpha_{yj}} v_j \Delta n_y \\ \Delta p_z &= \sum_{j=1}^{3} \frac{\mu}{\alpha_{zj}} v_j \Delta n_z \end{aligned} \tag{6.5}$$

where $1/\alpha_{ij}$ are the entries in the matrix D in Equation 6.1 (p. 263), v_j are the velocity components in the x, y, and z directions, and Δn_x, Δn_y, and Δn_z are the thicknesses of the medium in the x, y, and z directions.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Here, the thickness of the medium (Δn_x, Δn_y, or Δn_z) is the *actual* thickness of the porous region in your model. Therefore if the thicknesses used in your model differ from the actual thicknesses, you must make the adjustments in your inputs for $1/\ \alpha_{ij}$.

6.2.3.2.2. Inertial Losses in Porous Media

At high flow velocities, the constant C_2 in Equation 6.1 (p. 263) provides a correction for inertial losses in the porous medium. This constant can be viewed as a loss coefficient per unit length along the flow direction, thereby allowing the pressure drop to be specified as a function of dynamic head.

If you are modeling a perforated plate or tube bank, you can sometimes eliminate the permeability term and use the inertial loss term alone, yielding the following simplified form of the porous media equation:

$$\nabla p = -\sum_{j=1}^{3} C_{2_{ij}} \left(\frac{1}{2} \rho v_j |v| \right) \tag{6.6}$$

or when written in terms of the pressure drop in the x, y, z directions:

$$\begin{aligned} \Delta p_x &\approx \sum_{j=1}^{3} C_{2_{xj}} \Delta n_x \tfrac{1}{2} \rho v_j |v| \\ \Delta p_y &\approx \sum_{j=1}^{3} C_{2_{yj}} \Delta n_y \tfrac{1}{2} \rho v_j |v| \\ \Delta p_z &\approx \sum_{j=1}^{3} C_{2_{zj}} \Delta n_z \tfrac{1}{2} \rho v_j |v| \end{aligned} \tag{6.7}$$

Again, the thickness of the medium (Δn_x, Δn_y, or Δn_z) is the thickness you have defined in your model.

6.2.3.3. Relative Viscosity in Porous Media

For viscous flows, an effective viscosity, μ_e, is introduced to account for the effect of the porous medium on the diffusion term in the momentum equations:

$$\mu_e = \mu_r \mu \tag{6.8}$$

where μ is the fluid viscosity and μ_r is the relative viscosity. In Fluent the following sub-models are implemented to calculate the relative viscosity. The effective viscosity is then calculated from Equation 6.8 (p. 264).

Brinkman Correction[16] (p. 2887)

$$\mu_r = \left(1-\gamma\right)^{-2.5} \tag{6.9}$$

Einstein Formula[15] (p. 2887)

$$\mu_r = 1 + 2.5\left(1-\gamma\right) \tag{6.10}$$

Breugem Correlation [14] (p. 2887)

$$\mu_r = \begin{cases} \frac{1}{2}\left(\gamma - \frac{3}{7}\right) & \gamma \geq \frac{3}{7} \\ 0 & \gamma < \frac{3}{7} \end{cases} \tag{6.11}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.2.3.4. Treatment of the Energy Equation in Porous Media

ANSYS Fluent solves the standard energy transport equation (Equation 5.1 in the Theory Guide) in porous media regions with modifications to the conduction flux and the transient terms only.

6.2.3.4.1. Equilibrium Thermal Model Equations

For simulations in which the porous medium and fluid flow are assumed to be in thermal equilibrium, the conduction flux in the porous medium uses an effective conductivity and the transient term includes the thermal inertia of the solid region on the medium:

$$\begin{aligned}\frac{\partial}{\partial t}(\gamma\rho_{\mathrm{f}}E_{\mathrm{f}}+(1-\gamma)\rho_{\mathrm{s}}E_{\mathrm{s}}) \quad &+\nabla\cdot(\vec{v}(\rho_{\mathrm{f}}E_{\mathrm{f}}+p)) \\ &=S_{\mathrm{f}}^{h}+\nabla\cdot\left[k_{\mathrm{eff}}\nabla T-\left(\sum_{i}h_{i}J_{i}\right)+(\overline{\overline{\tau}}\cdot\vec{v})\right]\end{aligned} \tag{6.12}$$

where

E_{f} = total fluid energy

E_{s} = total solid medium energy

ρ_{f} = fluid density

ρ_{s} = solid medium density

γ = porosity of the medium

k_{eff} = effective thermal conductivity of the medium

S_{f}^{h} = fluid enthalpy source term

The effective thermal conductivity in the porous medium, k_{eff}, is computed by ANSYS Fluent as the volume average of the fluid conductivity and the solid conductivity:

$$k_{\mathrm{eff}}=\gamma k_{\mathrm{f}}+(1-\gamma)k_{\mathrm{s}} \tag{6.13}$$

where

k_{f} = fluid phase thermal conductivity (including the turbulent contribution, k_{t})

k_{s} = solid medium thermal conductivity

The fluid thermal conductivity k_{f} and the solid thermal conductivity k_{s} can be computed via user-defined functions.

The anisotropic effective thermal conductivity can also be specified via user-defined functions. In this case, the isotropic contributions from the fluid, γk_{f}, are added to the diagonal elements of the solid anisotropic thermal conductivity matrix.

6.2.3.4.2. Non-Equilibrium Thermal Model Equations

For simulations in which the porous medium and fluid flow are not assumed to be in thermal equilibrium, a dual cell approach is used. In such an approach, a solid zone that is spatially coincident with the porous fluid zone is defined, and this solid zone only interacts with the fluid with regard to heat transfer. The conservation equations for energy are solved separately for the fluid and solid zones. The conservation equation solved for the fluid zone is

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\frac{\partial}{\partial t}\left(\gamma\rho_{\mathrm{f}}E_{\mathrm{f}}\right)+\nabla\cdot\left(\vec{v}\left(\rho_{\mathrm{f}}E_{\mathrm{f}}+p\right)\right)=\nabla\cdot\left(\gamma k_{\mathrm{f}}\nabla T_{\mathrm{f}}-\left(\sum_{i}h_{i}J_{i}\right)+\left(\bar{\bar{\tau}}\cdot\vec{v}\right)\right)+S_{\mathrm{f}}^{h}+h_{\mathrm{fs}}A_{\mathrm{fs}}\left(T_{\mathrm{s}}-T_{\mathrm{f}}\right) \tag{6.14}$$

and the conservation equation solved for the solid zone is

$$\frac{\partial}{\partial t}\left(\left(1-\gamma\right)\rho_{\mathrm{s}}E_{\mathrm{s}}\right)=\nabla\cdot\left(\left(1-\gamma\right)k_{\mathrm{s}}\nabla T_{\mathrm{s}}\right)+S_{\mathrm{s}}^{h}+h_{\mathrm{fs}}A_{\mathrm{fs}}\left(T_{\mathrm{f}}-T_{\mathrm{s}}\right) \tag{6.15}$$

where

E_{f} = total fluid energy

E_{s} = total solid medium energy

ρ_{f} = fluid density

ρ_{s} = solid medium density

γ = porosity of the medium

k_{f} = fluid phase thermal conductivity (including the turbulent contribution, k_t)

k_{s} = solid medium thermal conductivity

h_{fs} = heat transfer coefficient for the fluid / solid interface

A_{fs} = interfacial area density, that is, the ratio of the area of the fluid / solid interface and the volume of the porous zone

T_{f} = temperature of the fluid

T_{s} = temperature of the solid medium

S_{f}^{h} = fluid enthalpy source term

S_{s}^{h} = solid enthalpy source term

The fluid thermal conductivity k_{f} and the solid thermal conductivity k_{s} can be computed via user-defined functions.

The source term due to the non-equilibrium thermal model is represented in Equation 6.14 (p. 266) and Equation 6.15 (p. 266) by $h_{\mathrm{fs}}A_{\mathrm{fs}}(T_{\mathrm{f}}-T_{\mathrm{s}})$ and $h_{\mathrm{fs}}A_{\mathrm{fs}}(T_{\mathrm{s}}-T_{\mathrm{f}})$, respectively.

6.2.3.5. Treatment of Turbulence in Porous Media

ANSYS Fluent will, by default, solve the standard conservation equations for turbulence quantities in the porous medium. In this default approach, turbulence in the medium is treated as though the solid medium has no effect on the turbulence generation or dissipation rates. This assumption may be reasonable if the medium's permeability is quite large and the geometric scale of the medium does not interact with the scale of the turbulent eddies. In other instances, however, you may want to suppress the effect of turbulence in the medium.

If you are using one of the turbulence models (with the exception of the Large Eddy Simulation (LES) model), you can suppress the effect of turbulence in a porous region by enabling the **Laminar Zone** option in the Fluid Dialog Box (p. 2369). Refer to Specifying a Laminar Zone (p. 255) for details about using the **Laminar Zone** option.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.2.3.6. Effect of Porosity on Transient Scalar Equations

For transient porous media calculations, the effect of porosity on the time-derivative terms is accounted for in all scalar transport equations and the continuity equation. When the effect of porosity is taken into account, the time-derivative term becomes $\frac{\partial}{\partial t}\left(\gamma\rho\phi\right)$, where ϕ is the scalar quantity (k, ε, and so on) and γ is the porosity.

The effect of porosity is enabled automatically for transient calculations, and the porosity is set to 1 by default.

6.2.3.7. User Inputs for Porous Media

When you are modeling a porous region, the additional inputs for the problem setup are as follows. Optional inputs are indicated as such.

1. Define the porous zone (as described in Defining the Porous Zone (p. 268)).
2. (optional) Define the porous velocity formulation in the **Cell Zone Conditions** task page (as described in Defining the Porous Velocity Formulation (p. 269)).
3. Identify the fluid material flowing through the porous medium (as described in Defining the Fluid Passing Through the Porous Medium (p. 269)).
4. Enable reactions for the porous zone, if appropriate, and select the reaction mechanism (as described in Enabling Reactions in a Porous Zone (p. 269)).
5. Enable the **Relative Velocity Resistance Formulation**. By default, this option is already enabled and takes the moving porous media into consideration (as described in Including the Relative Velocity Resistance Formulation (p. 269)).
6. Set the viscous resistance coefficients (D_{ij} in Equation 6.1 (p. 263), or $1/\alpha$ in Equation 6.2 (p. 263)) and the inertial resistance coefficients (C_{ij} in Equation 6.1 (p. 263), or C_2 in Equation 6.2 (p. 263)), and define the direction vectors for which they apply. Alternatively, specify the coefficients for the power-law model. (See Defining the Viscous and Inertial Resistance Coefficients (p. 270) for more details.)
7. Specify the porosity of the porous medium (as described in Defining Porosity (p. 277)).
8. (optional) Specify the settings for heat transfer (as described in Specifying the Heat Transfer Settings (p. 277)).
9. (optional) Specify a model for relative viscosity (as described in Specifying the Relative Viscosity (p. 280)).
10. (optional, Eulerian multiphase models only) Specify relative permeability (as described in Specifying the Relative Permeability (p. 280).
11. (optional) Set the volumetric heat generation rate in the non-solid portion of the porous medium (or any other sources, such as mass or momentum). (See Defining Sources (p. 283) for more details.)
12. (optional) Set any fixed values for solution variables in the fluid region (as described in Defining Fixed Values (p. 283).
13. Suppress the turbulent viscosity in the porous region, if appropriate. (See Suppressing the Turbulent Viscosity in the Porous Region (p. 283) for more details.)

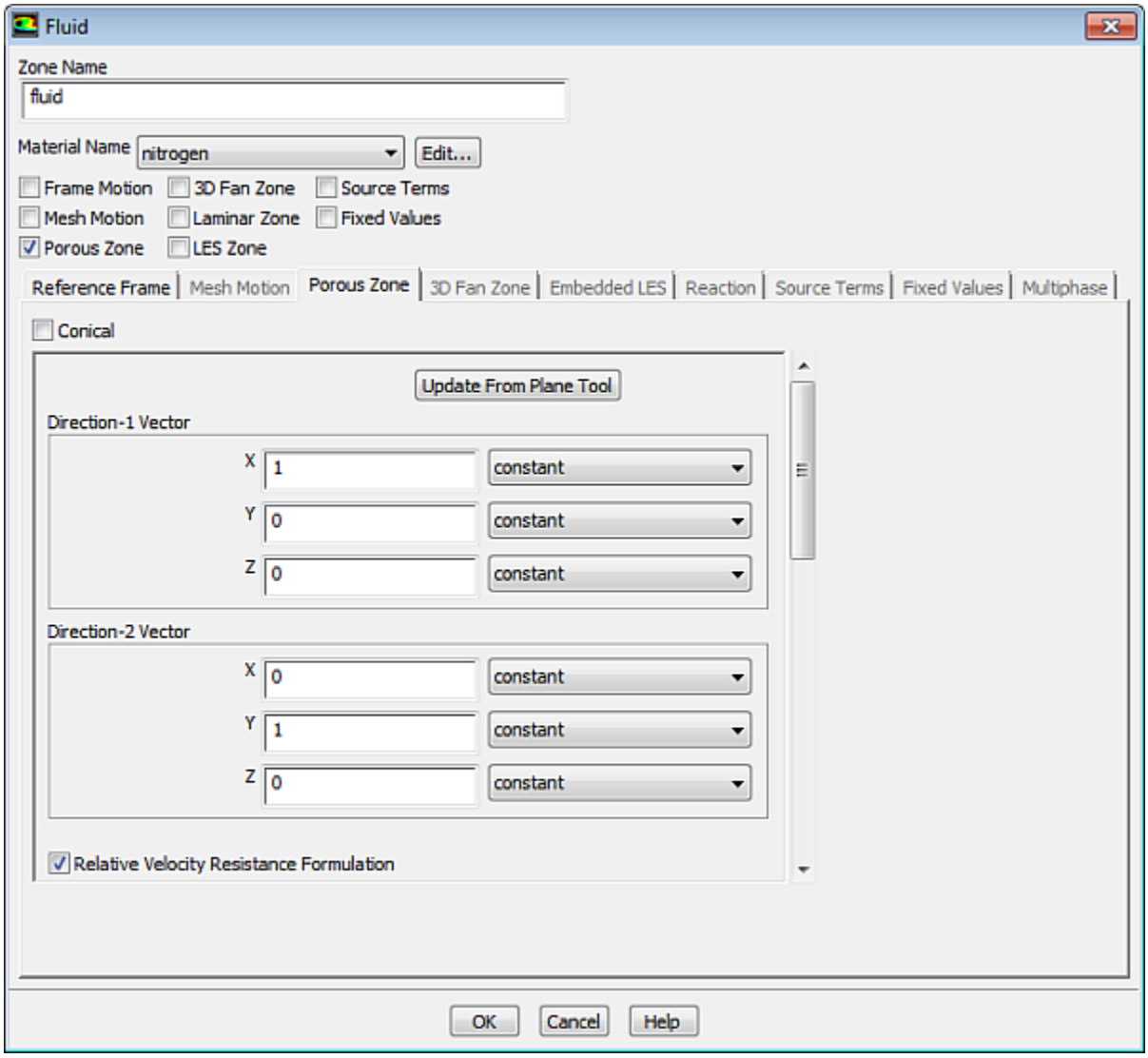
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14. Specify the rotation axis and/or zone motion, if relevant. (See Specifying the Rotation Axis and Defining Zone Motion (p. 283) for more details.)

Methods for determining the resistance coefficients and/or permeability are presented below. If you choose to use the power-law approximation of the porous-media momentum source term, you will enter the coefficients C_0 and C_1 in Equation 6.3 (p. 263) instead of the resistance coefficients and flow direction.

You will set all parameters for the porous medium in the Fluid Dialog Box (p. 2369) (Figure 6.12: The Fluid Dialog Box for a Porous Zone (p. 268)), which is opened from the **Cell Zone Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 241)).

Figure 6.12: The Fluid Dialog Box for a Porous Zone



6.2.3.7.1. Defining the Porous Zone

As mentioned in Overview (p. 237), a porous zone is modeled as a special type of fluid zone. To indicate that the fluid zone is a porous region, enable the **Porous Zone** option in the **Fluid** dialog box. The dialog box will expand to show the porous media inputs (as shown in Figure 6.12: The Fluid Dialog Box for a Porous Zone (p. 268)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In a coupled analysis involving Fluent and System Coupling, a porous jump boundary of zero-thickness next to the porous zone can be used to transfer data between the porous zone and the System Coupling system. For more about System Coupling and the variable transferred from a porous jump boundary, see Performing System Coupling Simulations Using Fluent in Workbench.

6.2.3.7.2. Defining the Porous Velocity Formulation

The **Cell Zone Conditions** task page contains a **Porous Formulation** region where you can instruct ANSYS Fluent to use either a superficial or physical velocity in the porous medium simulation. By default, the velocity is set to **Superficial Velocity**. For details about using the **Physical Velocity** formulation, see Modeling Porous Media Based on Physical Velocity (p. 283).

6.2.3.7.3. Defining the Fluid Passing Through the Porous Medium

To define the fluid that passes through the porous medium, select the appropriate fluid in the **Material Name** drop-down list in the Fluid Dialog Box (p. 2369). If you want to check or modify the properties of the selected material, you can click **Edit...** to open the **Edit Material** dialog box; this dialog box contains just the properties of the selected material, not the full contents of the standard **Create/Edit Materials** dialog box.

Important

If you are modeling species transport or multiphase flow, the **Material Name** list will not appear in the **Fluid** dialog box. For species calculations, the mixture material for all fluid/porous zones will be the material you specified in the Species Model Dialog Box (p. 2216). For multiphase flows, the materials are specified when you define the phases, as described in Defining the Phases for the VOF Model (p. 1436).

6.2.3.7.4. Enabling Reactions in a Porous Zone

If you are modeling species transport with reactions, you can enable reactions in a porous zone by turning on the **Reaction** option in the **Fluid** dialog box and selecting a mechanism in the **Reaction Mechanism** drop-down list.

If your mechanism contains wall surface reactions, you also must specify a value for the **Surface-to-Volume Ratio**. This value is the surface area of the pore walls per unit volume ($\frac{A}{V}$), and can be thought of as a measure of catalyst loading. With this value, ANSYS Fluent can calculate the total surface area on which the reaction takes place in each cell by multiplying $\frac{A}{V}$ by the volume of the cell. See Defining Zone-Based Reaction Mechanisms (p. 984) for details about defining reaction mechanisms. See Wall Surface Reactions and Chemical Vapor Deposition (p. 995) for details about wall surface reactions.

6.2.3.7.5. Including the Relative Velocity Resistance Formulation

For cases involving moving meshes, dynamic meshes or moving reference frames (MRF), the **Relative Velocity Resistance Formulation** option allows you to better predict porous media sources. The porous media source terms are calculated using relative velocities in the porous zone. For more information, see Momentum Equations for Porous Media (p. 262). The **Relative Velocity Resistance Formulation** option works well for cases with moving and stationary porous media. It is turned on by default.

Note the following limitation when using the **Relative Velocity Resistance Formulation** option:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The relative velocity resistance formulation is not supported with axisymmetric-swirl when there are non-zero swirl resistances.

Note

In ANSYS Fluent 6.3, this option was only supported for moving mesh and MRF (with the exception of power law for modeling porous sources). Currently, this option also supports dynamic mesh and takes care of proper treatment of porous sources using power law.

6.2.3.7.6. Defining the Viscous and Inertial Resistance Coefficients

The viscous and inertial resistance coefficients are both defined in the same manner. The basic approach for defining the coefficients using a Cartesian coordinate system is to define one direction vector in 2D or two direction vectors in 3D, and then specify the viscous and/or inertial resistance coefficients in each direction. In 2D, the second direction, which is not explicitly defined, is normal to the plane defined by the specified direction vector and the z direction vector. In 3D, the third direction is normal to the plane defined by the two specified direction vectors. For a 3D problem, the second direction must be normal to the first. If you fail to specify two normal directions, the solver will ensure that they are normal by ignoring any component of the second direction that is in the first direction. You should therefore be certain that the first direction is correctly specified.

You can also define the viscous and/or inertial resistance coefficients in each direction using a user-defined function (UDF). The user-defined options become available in the corresponding drop-down list when the UDF has been created and loaded into ANSYS Fluent. Note that the coefficients defined in the UDF must utilize the `DEFINE_PROFILE` macro. For more information on creating and using user-defined function, see the Fluent Customization Manual.

If you are modeling axisymmetric swirling flows, you can specify an additional direction component for the viscous and/or inertial resistance coefficients. This direction component is always tangential to the other two specified directions. This option is available for both density-based and pressure-based solvers.

In 3D, it is also possible to define the coefficients using a conical (or cylindrical) coordinate system, as described below.

Important

Note that the viscous and inertial resistance coefficients are generally based on the superficial velocity of the fluid in the porous media.

The procedure for defining resistance coefficients is as follows:

1. Define the direction vectors.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To use a Cartesian coordinate system, simply specify the **Direction-1 Vector** and, for 3D, the **Direction-2 Vector**. The unspecified direction will be determined as described above. These direction vectors correspond to the principle axes of the porous media.

  Note

  The units for the inertial resistance coefficients (**Direction-1 Vector** and **Direction-2 Vector**) is the inverse of length. Should you want to define different units, you can do so by opening the **Units** dialog box and selecting **resistance** from the **Quantities** list.

  For some problems in which the principal axes of the porous medium are not aligned with the coordinate axes of the domain, you may not know a priori the direction vectors of the porous medium. In such cases, the plane tool in 3D (or the line tool in 2D) can help you to determine these direction vectors.

  a. "Snap" the plane tool (or the line tool) onto the boundary of the porous region. (Follow the instructions in Using the Plane Tool (p. 1724) or Using the Line Tool (p. 1720) for initializing the tool to a position on an existing surface.)

  b. Rotate the axes of the tool appropriately until they are aligned with the porous medium.

  c. Once the axes are aligned, click the **Update From Plane Tool** or **Update From Line Tool** button in the **Fluid** dialog box. ANSYS Fluent will automatically set the **Direction-1 Vector** to the direction of the red arrow of the tool, and (in 3D) the **Direction-2 Vector** to the direction of the green arrow.

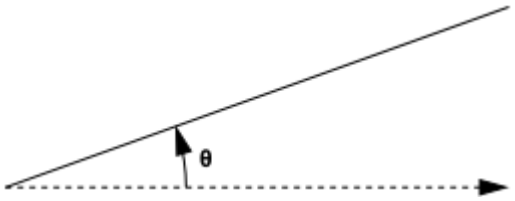
- To use a conical coordinate system (for example, for an annular, conical filter element), follow the steps below. This option is available only in 3D cases.

  a. Turn on the **Conical** option.

  b. Set the **Cone Half Angle** (the angle between the cone's axis and its surface, shown in Figure 6.13: Cone Half Angle (p. 271)). To use a cylindrical coordinate system, set the **Cone Half Angle** to 0.

  c. Specify the **Cone Axis Vector** and **Point on Cone Axis**. The cone axis is specified as being in the direction of the **Cone Axis Vector** (unit vector), and passing through the **Point on Cone Axis**. The cone axis may or may not pass through the origin of the coordinate system.

  Figure 6.13: Cone Half Angle

  For some problems in which the axis of the conical filter element is not aligned with the coordinate axes of the domain, you may not know a priori the direction vector of the cone axis and coordinates of a point on the cone axis. In such cases, the plane tool can help you to determine the cone axis vector and point coordinates. One method is as follows:

  a. Select a boundary zone of the conical filter element that is normal to the cone axis vector in the drop-down list next to the **Snap to Zone** button.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Click the **Snap to Zone** button. ANSYS Fluent will automatically "snap" the plane tool onto the boundary. It will also set the **Cone Axis Vector** and the **Point on Cone Axis**. (Note that you will still have to set the **Cone Half Angle** yourself.)

An alternate method is as follows:

a. "Snap" the plane tool onto the boundary of the porous region. (Follow the instructions in Using the Plane Tool (p. 1724) for initializing the tool to a position on an existing surface.)

b. Rotate and translate the axes of the tool appropriately until the red arrow of the tool is pointing in the direction of the cone axis vector and the origin of the tool is on the cone axis.

c. Once the axes and origin of the tool are aligned, click the **Update From Plane Tool** button in the **Fluid** dialog box. ANSYS Fluent will automatically set the **Cone Axis Vector** and the **Point on Cone Axis**. (Note that you will still have to set the **Cone Half Angle** yourself.)

2. Under **Viscous Resistance**, specify the viscous resistance coefficient $1/\alpha$ in each direction.

Under **Inertial Resistance**, specify the inertial resistance coefficient C_2 in each direction. (You can scroll down with the scroll bar to view these inputs.)

For porous media cases containing highly anisotropic inertial resistances, enable **Alternative Formulation** under **Inertial Resistance**. The **Alternative Formulation** option provides better stability to the calculation when your porous medium is anisotropic. The pressure loss through the medium depends on the magnitude of the velocity vector of the i^{th} component in the medium. Using the formulation of Equation 6.6 (p. 264) yields the expression below:

$$S_i = \frac{1}{2}\rho C_i |v_i| v_i \tag{6.16}$$

Whether or not you use the **Alternative Formulation** option depends on how well you can fit your experimentally determined pressure drop data to the ANSYS Fluent model. For example, if the flow through the medium is aligned with the mesh in your ANSYS Fluent model, then it will not make a difference whether or not you use the formulation. In general, however, the default and the **Alternative Formulation** are not equivalent, and solution differences can occur when switching between the two options.

For more information about simulations involving highly anisotropic porous media, see Solution Strategies for Porous Media (p. 286).

Important

Note that the alternative formulation is compatible only with the pressure-based solver.

If you are using the **Conical** specification method, **Direction-1** is the tangential direction of the cone, **Direction-2** is the normal to the cone surface (radial (r) direction for a cylinder), and **Direction-3** is the circumferential (θ) direction.

In 3D there are three possible categories of coefficients, and in 2D there are two:

- In the isotropic case, the resistance coefficients in all directions are the same (for example, a sponge). For an isotropic case, you must explicitly set the resistance coefficients in each direction to the same value.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When (in 3D) the coefficients in two directions are the same and those in the third direction are different or (in 2D) the coefficients in the two directions are different, you must be careful to specify the coefficients properly for each direction. For example, if you had a porous region consisting of cylindrical straws with small holes in them positioned parallel to the flow direction, the flow would pass easily through the straws, but the flow in the other two directions (through the small holes) would be very little. If you had a plane of flat plates perpendicular to the flow direction, the flow would not pass through them at all; it would instead move in the other two directions.

- In 3D the third possible case is one in which all three coefficients are different. For example, if the porous region consisted of a plane of irregularly-spaced objects (for example, pins), the movement of flow between the blockages would be different in each direction. You would therefore need to specify different coefficients in each direction.

Methods for deriving viscous and inertial loss coefficients are described in the sections that follow.

6.2.3.7.7. Deriving Porous Media Inputs Based on Superficial Velocity, Using a Known Pressure Loss

When you use the porous media model, you must remember that the porous cells in ANSYS Fluent are *100% open*, and that the values that you specify for $1/\ \alpha_{ij}$ and/or $C_{2_{ij}}$ must be based on this assumption. Suppose, however, that you know how the pressure drop varies with the velocity through the actual device, which is only partially open to flow. The following exercise is designed to show you how to compute a value for C_2 which is appropriate for the ANSYS Fluent model.

Consider a perforated plate that has 25% area open to flow. The pressure drop through the plate is known to be 0.5 times the dynamic head in the plate. The loss factor, K_L, defined as

$$\Delta p = K_L\left(\frac{1}{2}\rho v_{25\%open}^2\right) \tag{6.17}$$

is therefore 0.5, based on the actual fluid velocity in the plate, that is, the velocity through the 25% open area. To compute an appropriate value for C_2, note that in the ANSYS Fluent model:

1. The velocity through the perforated plate assumes that the plate is 100% open.

2. The loss coefficient must be converted into dynamic head loss per unit length of the porous region.

Noting item 1, the first step is to compute an adjusted loss factor, $K_L^{'}$, which would be based on the velocity of a 100% open area:

$$\Delta p = K_L^{'}\left(\frac{1}{2}\rho v_{100\%open}^2\right) \tag{6.18}$$

or, noting that for the same flow rate, $v_{25\%open} = 4 \times v_{100\%open}$,

$$K_L^{'} = K_L \times \frac{v_{25\%open}^2}{v_{100\%open}^2} = 0.5 \times \left(\frac{4}{1}\right)^2 = 8 \tag{6.19}$$

The adjusted loss factor has a value of 8. Noting item 2, you must now convert this into a loss coefficient per unit thickness of the perforated plate. Assume that the plate has a thickness of 1.0 mm (10^{-3} m). The inertial loss factor would then be

$$C_2 = \frac{K_{K_L^{'}}}{thickness} = \frac{8}{10^{-3}} = 8000 m^{-1} \tag{6.20}$$

Note that, for anisotropic media, this information must be computed for each of the 2 (or 3) coordinate directions.

6.2.3.7.8. Using the Ergun Equation to Derive Porous Media Inputs for a Packed Bed

As a second example, consider the modeling of a packed bed. In turbulent flows, packed beds are modeled using both a permeability and an inertial loss coefficient. One technique for deriving the appropriate constants involves the use of the Ergun equation [28] (p. 2888), a semi-empirical correlation applicable over a wide range of Reynolds numbers and for many types of packing:

$$\frac{|\Delta p|}{L} = \frac{150\mu}{D_p^2}\frac{\left(1-\varepsilon\right)^2}{\varepsilon^3}v_\infty + \frac{1.75\rho}{D_p}\frac{\left(1-\varepsilon\right)}{\varepsilon^3}v_\infty^2 \tag{6.21}$$

When modeling laminar flow through a packed bed, the second term in the above equation may be dropped, resulting in the Blake-Kozeny equation [28] (p. 2888):

$$\frac{|\Delta p|}{L} = \frac{150\mu}{D_p^2}\frac{\left(1-\varepsilon\right)^2}{\varepsilon^3}v_\infty \tag{6.22}$$

In these equations, μ is the viscosity, D_p is the mean particle diameter, L is the bed depth, and ε is the void fraction, defined as the volume of voids divided by the volume of the packed bed region. Comparing Equation 6.4 (p. 263) and Equation 6.6 (p. 264) with Equation 6.21 (p. 274), the permeability and inertial loss coefficient in each component direction may be identified as

$$\alpha = \frac{D_p^2}{150}\frac{\varepsilon^3}{\left(1-\varepsilon\right)^2} \tag{6.23}$$

and

$$C_2 = \frac{3.5}{D_p}\frac{\left(1-\varepsilon\right)}{\varepsilon^3} \tag{6.24}$$

6.2.3.7.9. Using an Empirical Equation to Derive Porous Media Inputs for Turbulent Flow Through a Perforated Plate

As a third example we will take the equation of Van Winkle et al. [83] (p. 2891) [110] (p. 2893) and show how porous media inputs can be calculated for pressure loss through a perforated plate with square-edged holes.

The expression, which is claimed by the authors to apply for turbulent flow through square-edged holes on an equilateral triangular spacing, is

$$\dot{m} = CA_f\sqrt{\left(2\rho\Delta p\right) / \left(1-\left(A_f / A_p\right)^2\right)} \tag{6.25}$$

where

$\dot{m}$ = mass flow rate through the plate

A_f = the free area or total area of the holes

A_p = the area of the plate (solid and holes)

C = a coefficient that has been tabulated for various Reynolds-number ranges and for various D / t

D / t = the ratio of hole diameter to plate thickness

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

for $t/D>1.6$ and for $Re>4000$ the coefficient C takes a value of approximately 0.98, where the Reynolds number is based on hole diameter and velocity in the holes.

Rearranging Equation 6.25 (p. 274), making use of the relationship

$$\dot{m}=\rho v A_p \tag{6.26}$$

and dividing by the plate thickness, $\Delta x=t$, we obtain

$$\frac{\Delta p}{\Delta x}=\left(\frac{1}{2}\rho v^2\right)\frac{1}{C^2}\frac{\left(A_p/A_f\right)^2-1}{t} \tag{6.27}$$

where v is the superficial velocity (not the velocity in the holes). Comparing with Equation 6.6 (p. 264) is seen that, for the direction normal to the plate, the constant C_2 can be calculated from

$$C_2=\frac{1}{C^2}\frac{\left(A_p/A_f\right)^2-1}{t} \tag{6.28}$$

6.2.3.7.10. Using Tabulated Data to Derive Porous Media Inputs for Laminar Flow Through a Fibrous Mat

Consider the problem of laminar flow through a mat or filter pad which is made up of randomly-oriented fibers of glass wool. As an alternative to the Blake-Kozeny equation (Equation 6.22 (p. 274)) we might choose to employ tabulated experimental data. Such data is available for many types of fiber [46] (p. 2889).

Volume Fraction of Solid Material	Dimensionless Permeability B of Glass Wool
0.262	0.25
0.258	0.26
0.221	0.40
0.218	0.41
0.172	0.80

where $B=\alpha/a^2$ and a is the fiber diameter. α, for use in Equation 6.4 (p. 263), is easily computed for a given fiber diameter and volume fraction.

6.2.3.7.11. Deriving the Porous Coefficients Based on Experimental Pressure and Velocity Data

Experimental data that is available in the form of pressure drop against velocity through the porous component, can be extrapolated to determine the coefficients for the porous media. To effect a pressure drop across a porous medium of thickness, Δn, the coefficients of the porous media are determined in the manner described below.

If the experimental data is:

Velocity (m/s)	Pressure Drop (Pa)
20.0	197.8
50.0	948.1
80.0	2102.5
110.0	3832.9

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

then an xy curve can be plotted to create a trendline through these points yielding the following equation

$$\Delta p = 0.27394 v^2 + 4.68816 v \tag{6.29}$$

where Δp is the pressure drop and v is the velocity.

Important

Although the best fit curve may yield negative coefficients, it should be avoided when using the porous media model in ANSYS Fluent.

Note that a simplified version of the momentum equation, relating the pressure drop to the source term, can be expressed as

$$\nabla p = S_i \tag{6.30}$$

or

$$\Delta p = -S_i \Delta n \tag{6.31}$$

Hence, comparing Equation 6.29 (p. 276) to Equation 6.2 (p. 263), yields the following curve coefficients:

$$0.27394 = C_2 \frac{1}{2} \rho \Delta n \tag{6.32}$$

with $\rho = 1.225$ kg/m^3, and a porous media thickness, Δn, assumed to be 1 m in this example, the inertial resistance factor, $C_2 = 0.447$.

Likewise,

$$4.68816 = \frac{\mu}{\alpha} \Delta n \tag{6.33}$$

with $\mu = 1.7894 \times 10^{-5}$, the viscous inertial resistance factor, $\frac{1}{\alpha} = 261996$.

Important

Note that this same technique can be applied to the porous jump boundary condition. Similar to the case of the porous media, you have to take into account the thickness of the medium Δn. Your experimental data can be plotted in an xy curve, yielding an equation that is equivalent to Equation 6.140 (p. 403). From there, you can determine the permeability α and the pressure jump coefficient C_2.

6.2.3.7.12. Using the Power-Law Model

If you choose to use the power-law approximation of the porous-media momentum source term (Equation 6.3 (p. 263)), the only inputs required are the coefficients C_0 and C_1. Under **Power Law Model** in the **Fluid** dialog box, enter the values for **C0** and **C1**. Note that the power-law model can be used in conjunction with the Darcy and inertia models.

C0 must be in SI units, consistent with the value of **C1**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.2.3.7.13. Defining Porosity

To define the porosity, scroll down below the resistance inputs in the **Fluid** dialog box, and set the **Porosity** under **Fluid Porosity**. Note that the value of the **Porosity** must be in the range of `0`–`1`; furthermore, the lower and upper limits (that is, `0` and `1`, respectively) are not allowed for the non-equilibrium thermal model.

You can also define the porosity using a user-defined function (UDF). The user-defined option becomes available in the corresponding drop-down list when the UDF has been created and loaded into ANSYS Fluent. Note that the porosity defined in the UDF must utilize the `DEFINE_PROFILE` macro. For more information on creating and using user-defined functions, see the separate Fluent Customization Manual.

The porosity, γ, is the volume fraction of fluid within the porous region (that is, the open volume fraction of the medium). The porosity is used in the prediction of heat transfer in the medium, as described in Treatment of the Energy Equation in Porous Media (p. 265), and in the time-derivative term in the scalar transport equations for unsteady flow, as described in Effect of Porosity on Transient Scalar Equations (p. 267). It also impacts the calculation of reaction source terms and body forces in the medium. These sources will be proportional to the fluid volume in the medium. If you want to represent the medium as completely open (no effect of the solid medium), you should set the porosity equal to 1.0 (the default). When the porosity is equal to 1.0, the solid portion of the medium will have no impact on heat transfer or thermal/reaction source terms in the medium.

6.2.3.7.14. Specifying the Heat Transfer Settings

You can model heat transfer in the porous material, with or without the assumption of thermal equilibrium between the medium and the fluid flow. Note that heat transfer is not available for inviscid flow.

6.2.3.7.14.1. Equilibrium Thermal Model

To specify that the porous medium and the fluid flow are in thermal equilibrium, scroll down below the resistance inputs in the **Fluid** dialog box to the **Heat Transfer Settings** group box and select **Equilibrium** from the **Thermal Model** list (this is the default selection). Then you must specify the material contained in the porous medium by selecting the appropriate solid in the **Solid Material Name** drop-down list.

If you want to check or modify the properties of the selected material, you can click **Edit...** to open the **Edit Material** dialog box; this dialog box contains just the properties of the selected material, not the full contents of the standard **Create/Edit Materials** dialog box. You can define the **Thermal Conductivity** of the porous material using a user-defined function (UDF), in order to define a non-isotropic thermal conductivity. The user-defined option becomes available in the corresponding drop-down list when the UDF has been created and loaded into ANSYS Fluent. Note that the non-isotropic thermal conductivity defined in the UDF must utilize the `DEFINE_PROPERTY` macro. For more information on creating and using user-defined function, see the separate Fluent Customization Manual.

6.2.3.7.14.2. Non-Equilibrium Thermal Model

Note

The non-equilibrium thermal model is incompatible with the density based solver.

The assumption of thermal equilibrium between the solid medium and the fluid flow is not appropriate for all simulations, as the presence of different geometric length scales (for example, pore sizes) and

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

physical properties of solid and liquid phases may result in local temperature differences between the phases. Examples of when a non-equilibrium thermal model may be suitable include the simulation of light-off for exhaust after-treatment, fuel cells, catalytic converters, and so on.

The non-equilibrium thermal model available for porous media in ANSYS Fluent is based on a dual cell approach. This approach is referred to as "dual cell" because it involves a second solid cell zone that overlaps (that is, is spatially coincident with) the porous fluid zone; the two zones are solved simultaneously and are coupled only through heat transfer. ANSYS Fluent can automatically create a solid zone for you that is a duplicate of the porous fluid zone; otherwise, you can create a duplicate cell zone manually using the `mesh/modify-zones/copy-move-cell-zone` text command before you enable the non-equilibrium thermal model, or make sure that the mesh you read contains two cell zones in the region where the porous medium will be defined. You should ensure that the duplicate zone is defined as a solid zone, and that the two zones have similar levels of mesh refinement (as one-to-one mapping will be employed between the cell centroids of the fluid zone and the solid zone).

Important

When setting up both the porous fluid zone and the overlapping solid zone, note that the non-equilibrium thermal model is not supported for zones that undergo (or are adjacent to zones that undergo) changes to the geometry or mesh, such as dynamic mesh zones or zones that undergo adaption, utilize the mesh morpher/optimizer, or are involved in fluid-structure interaction (FSI) applications.

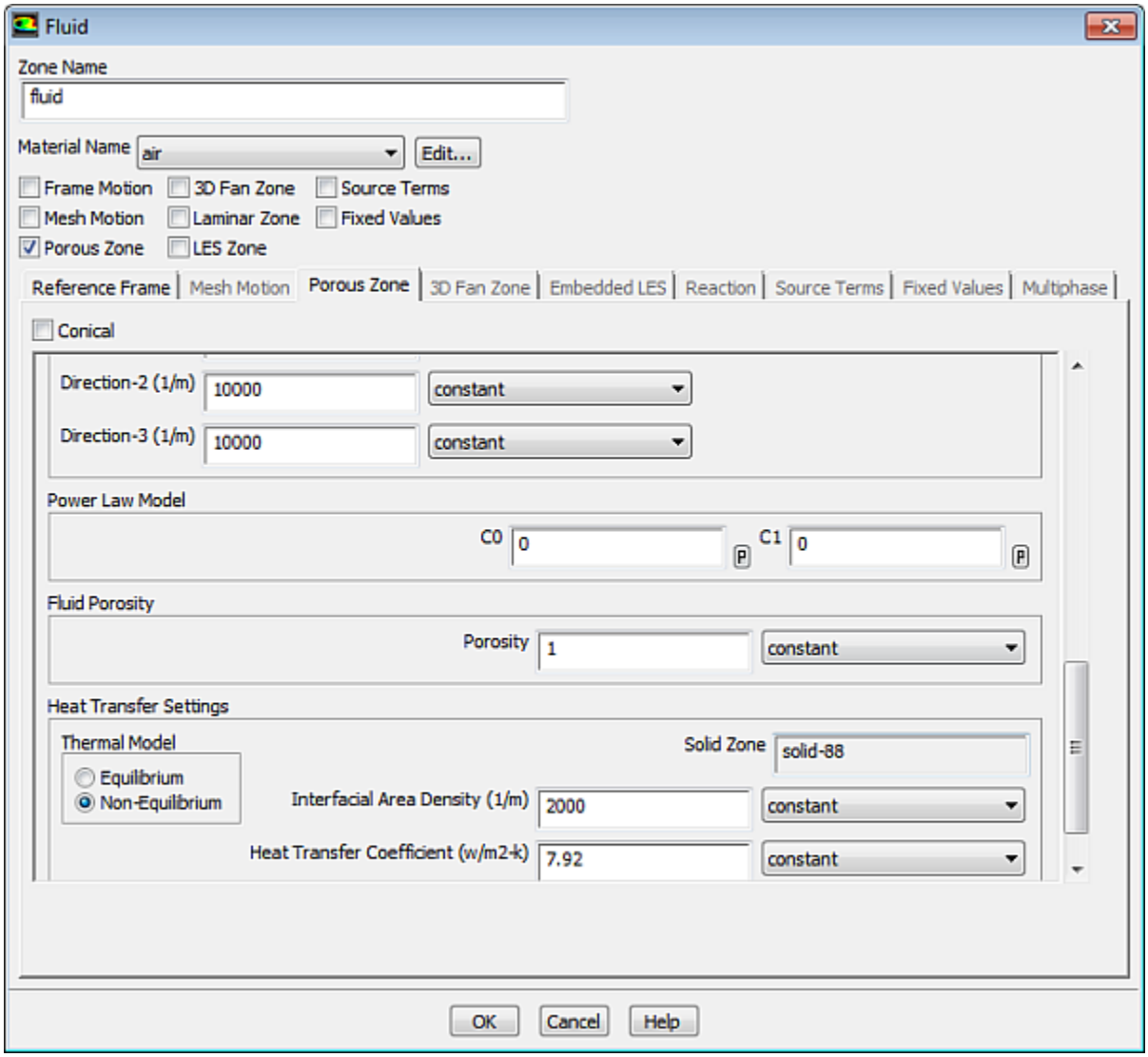
The instructions that follow assume you have already performed steps 1–7 in User Inputs for Porous Media (p. 267). Before you enable the non-equilibrium thermal model, click **OK** in the **Fluid** dialog box, in order to save your settings in the **Porous Zone** tab; note that if you do not save your settings, they will be reset to the default values when you enable the non-equilibrium thermal model. Then reopen the **Fluid** dialog box and scroll down below the resistance inputs in the **Porous Zone** tab. Select **Non-Equilibrium** from the **Thermal Model** list in the **Heat Transfer Settings** group box (see Figure 6.14: The Heat Transfer Settings Group Box of the Fluid Dialog Box (p. 279)); this will enable the dual cell approach and display the associated GUI input controls. If a solid zone that is spatially coincident with the porous fluid zone does not already exist, use the **Question** dialog box that opens to automatically create one. The name of the newly created solid cell zone will then be displayed in the **Solid Zone** text box of the **Fluid** dialog box.

Important

Note that the **Non-Equilibrium** option button is not available if a radiation and/or multiphase model is enabled, as such a combination is not supported.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.14: The Heat Transfer Settings Group Box of the Fluid Dialog Box



Next, define A_{fs} and h_{fs} (as described in Non-Equilibrium Thermal Model Equations (p. 265)) for **Interfacial Area Density** and **Heat Transfer Coefficient**, respectively; you can define these settings as a **Constant**, a **New Input Parameter...**, or as a user-defined function. The **user-defined** option becomes available in the corresponding drop-down list when the UDF has been created and loaded into ANSYS Fluent. Note that the UDF must utilize the `DEFINE_PROFILE` macro. For more information on creating and using user-defined functions, see the separate Fluent Customization Manual.

When you are finished setting up the porous fluid zone, you should verify that the solid zone created in the previous step has the appropriate settings. Note the name of the zone displayed in the **Solid Zone** text box of the **Heat Transfer Settings** group box (see Figure 6.14: The Heat Transfer Settings Group Box of the Fluid Dialog Box (p. 279)), and then double-click that zone in the **Zone** list of the **Cell Zone Conditions** task page to open the **Solid** dialog box. Then, verify that an appropriate selection is made for **Material Name**. If you want to check or modify the properties of the selected material, you can click **Edit...** to open the **Edit Material** dialog box; this dialog box contains just the properties of the selected material, not the full contents of the standard **Create/Edit Materials** dialog box. You can define the **Thermal Conductivity** of the solid material using a user-defined function (UDF), in order to define a non-isotropic thermal conductivity. The user-defined option becomes available in the corresponding drop-down list when the UDF has been created and loaded into ANSYS Fluent. Note that the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

non-isotropic thermal conductivity defined in the UDF must utilize the DEFINE_PROPERTY macro. For more information on creating and using user-defined functions, see the separate Fluent Customization Manual.

Note that when postprocessing simulations that utilize the non-equilibrium thermal model, you can display the energy source due to the temperature difference between the fluid and solid zones using the **Non-Equilibrium Thermal Model Source** variable in the **Temperature...** category. Furthermore, attention must be paid to the fact that there are overlapping cell zones (that is, a fluid and a solid zone) in the porous region. For example:

- When creating an isosurface and making selections from the **From Zones** selection list of the **Iso-Surface** dialog box, you should never select both the porous fluid zone and the overlapping solid zone at the same time. Note that if no selections are made in this list, then all the cell zones are selected.
- When making selections from the **Surfaces** list of the **Contours** dialog box, you should not select a surface that is associated with the porous fluid zone and a surface associated with the overlapping solid zone at the same time, if those surfaces are spatially coincident.
- When postprocessing the thermal conductivity property in the porous region, the value that is reported for the fluid zone is the thermal conductivity of the fluid material scaled by the porosity (γk_{f}) and for the overlapping solid zone the value reported is the thermal conductivity of the solid zone scaled by the complement of the porosity ($(1-\gamma)k_{\mathrm{s}}$); where γ is the porosity, k_{f} is the fluid phase thermal conductivity (including the turbulent contribution, k_{t}), and k_{s} is the solid medium thermal conductivity.

6.2.3.7.15. Specifying the Relative Viscosity

The relative viscosity can be used to account for the effect of the porous media on the diffusion term in the momentum equation. By default, the relative viscosity is set to a constant value of 1 (that is, the effective viscosity is equal to the molecular viscosity). You can choose to model the relative viscosity using one of the models described in Relative Viscosity in Porous Media (p. 264), or you can specify a user-defined function using a DEFINE_PROPERTY macro.

6.2.3.7.16. Specifying the Relative Permeability

For two-phase flow through a porous medium, the flow rate of each phase is dictated by the portion of the total cross-sectional area A_q that is available for the phase to flow and the permeability of the medium. Since the portion of the total cross-sectional area A_q is difficult to determine, the reservoir engineers use the concept of effective permeability:

$$K_{e,q}=KA_q \tag{6.34}$$

The effective permeability to a phase $K_{e,q}$ is the ability of a porous medium to transmit that phase when its saturation in the porous medium is less than 100%. The effective permeability to a phase depends on the permeability of the medium K, the phase saturation, the saturations of other phases present in the medium, the wettability preference of the medium, and its pore characteristics and size distribution.

A typical reservoir displays a range of effective permeabilities and permeability variations. To overcome this problem, the concept of the relative permeability is used:

$$K_{r,q}=K_{e,q}/K_b \tag{6.35}$$

The relative permeability $K_{r,q}$, therefore, is a ratio of the effective permeability $K_{e,q}$ to some base permeability K_b. It is a dimensionless measure of the effective permeability and varies with the wetting phase saturation. There is no consensus for the choice of base permeability; however, for oil-water

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

systems, K_b is $K_{e,oil}$ at irreducible water saturation for the oil phase, and K_b is $K_{e,water}$ at residual oil saturation for the water phase.

Reservoir engineers often resort to empirical models or measurement curves to determine the relative and, therefore, the effective permeabilities. In Fluent, the Corey power law model [17] (p. 2887) is available to account for the relative permeability for the two-phase flow through porous media. The Corey correlations of the relative permeability can be represented as follows:

$$K_{r,q}=K_{r,q}^{0}\tilde{S}_{q}^{nq} \tag{6.36}$$

where:

$K_{r,q}^{0}$ is the reference phase relative permeability

$\tilde{S}_q$ is the normalized saturation in the qth phase

nq is the Corey exponent for the qth phase

In Fluent the phase saturation is the phase volume fraction. Thus $\tilde{S}_q$ can be expressed as:

$$\tilde{S}_{q}=\tilde{\alpha}_{q}=\frac{\alpha_q-\alpha_{qr}}{1-\sum_{q=0}^{n-1}\alpha_{qr}} \tag{6.37}$$

where α_{qr} is the volume fraction residual in the qth phase.

For the two-phase oil-water flow example, the relative permeabilities can be obtained from:

$$K_{r,w}=K_{r,w}^{0}\tilde{\alpha}_{w}^{nw}$$
$$K_{r,o}=K_{r,o}^{0}\left(1-\tilde{\alpha}_{w}\right)^{no} \tag{6.38}$$

This requires the specification of 6 parameters:

$K_{r,w}^{0}$ and $K_{r,o}^{0}$	the reference phase relative permeabilities for the two phases
α_{wr} and α_{or}	the residual volume fractions for the two phases
nw and no	the Corey exponents for the two phases

The Corey correlation is often used for approximation of the relative permeabilities, particularly for water flooding type scenarios. The relative permeability curves of the oil (or the non-wetting phase) is usually an S-shaped curve and of the water (or the wetting phase) is a concave upward curve

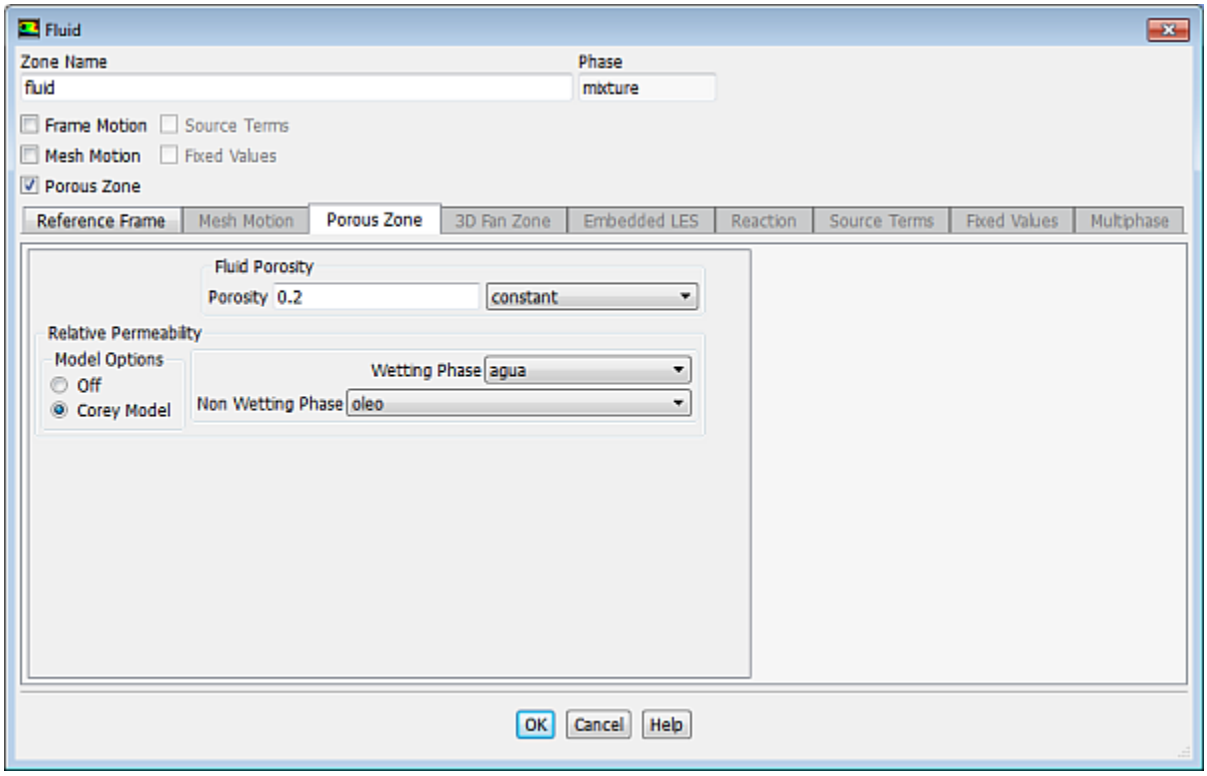
To model the relative permeability in ANSYS Fluent:

1. Set parameters for the **mixture** phase (**mixture** is selected for **Phase** in the **Cell Zone Conditions** task page).

   a. In the **Fluid** dialog box, in the **Porous Zone** tab, under **Relative Permeability**, select **Corey Model**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.15: The Fluid Dialog Box: Relative Permeability



b. From the drop-down lists, select **Wetting Phase** and **Non Wetting Phase**.

> **Important**
>
> Although Equation 6.35 (p. 280) through Equation 6.38 (p. 281) are valid for any given phase, care must be exercised in defining the wetting and non-wetting phases.

2. Set conditions for the wetting phase (the wetting phase is selected from the **Phase** drop-down list in the **Cell Zone Conditions** task page). In the **Fluid** dialog box, in the **Porous Zone** tab, specify the following parameters in the **Wetting Phase Relative Permeability** group:

   Reference Relative Permeability

   $K^0_{r,w}$ in Equation 6.38 (p. 281)

   Corey Exponent

   α_{wr} in Equation 6.38 (p. 281)

   Residual Saturation

   nw in Equation 6.38 (p. 281)

3. Similarly, set conditions for the non-wetting phase (the non-wetting phase is selected from the **Phase** drop-down list in the **Cell Zone Conditions** task page). In the **Fluid** dialog box, in the **Porous Zone** tab, specify the following parameters in the **Non Wetting Phase Relative Permeability** group:

   Reference Relative Permeability

   $K^0_{r,o}$ in Equation 6.38 (p. 281)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Corey Exponent
α_{or} in Equation 6.38 (p. 281)

Residual Saturation
no in Equation 6.38 (p. 281)

6.2.3.7.17. Defining Sources

If you want to include effects of the heat generated by the porous medium in the energy equation, enable the **Source Terms** option and set a nonzero **Energy** source. The solver will compute the heat generated by the porous region by multiplying this value by the total volume of the cells comprising the porous zone. You may also define sources of mass, momentum, turbulence, species, or other scalar quantities, as described in Defining Mass, Momentum, Energy, and Other Sources (p. 297).

6.2.3.7.18. Defining Fixed Values

If you want to fix the value of one or more variables in the fluid region of the zone, rather than computing them during the calculation, you can do so by enabling the **Fixed Values** option. See Fixing the Values of Variables (p. 292) for details.

6.2.3.7.19. Suppressing the Turbulent Viscosity in the Porous Region

As discussed in Treatment of Turbulence in Porous Media (p. 266), turbulence will be computed in the porous region just as in the bulk fluid flow. If you are using one of the turbulence models (other than the Large Eddy Simulation model) and you want the turbulence generation to be zero in the porous zone, turn on the **Laminar Zone** option in the **Fluid** dialog box. Refer to Specifying a Laminar Zone (p. 255) for more information about suppressing turbulence generation.

6.2.3.7.20. Specifying the Rotation Axis and Defining Zone Motion

Inputs for the rotation axis and zone motion are the same as for a standard fluid zone. See Inputs for Fluid Zones (p. 253) for details.

6.2.3.8. Modeling Porous Media Based on Physical Velocity

As stated in Limitations and Assumptions of the Porous Media Model (p. 261), by default ANSYS Fluent calculates the superficial velocity based on volumetric flow rate. The superficial velocity in the governing equations can be represented as

$$\vec{v}_{superficial} = \gamma \vec{v}_{physical} \tag{6.39}$$

where γ is the porosity of the media defined as the ratio of the volume occupied by the fluid to the total volume.

The superficial velocity values within the porous region remain the same as those outside of the porous region, and porosity is not taken into account in the differential terms of the transport equations. This limits the accuracy of the porous model in cases where there should be an increase in velocity throughout the porous region, and does not yield accurate results when velocity values and gradients are important. For more accurate simulations of porous media flows, it becomes necessary to solve for the true, or physical, velocity throughout the flowfield rather than the superficial velocity, as well as to include porosity in all terms of the transport equations.

ANSYS Fluent allows the calculation of the physical velocity using the **Porous Formulation**, available in the **Cell Zone Conditions** task page. By default, the **Superficial Velocity** option is turned on.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.2.3.8.1. Single Phase Porous Media

Using the physical velocity formulation, and assuming a general scalar ϕ, the governing equation in an isotropic porous media has the following form:

$$\frac{\partial(\gamma\rho\phi)}{\partial t}+\nabla\cdot(\gamma\rho\vec{v}\phi)=\nabla\cdot(\gamma\Gamma\nabla\phi)+\gamma S_{\phi} \tag{6.40}$$

Assuming isotropic porosity and single phase flow, the volume-averaged mass and momentum conservation equations are as follows:

$$\frac{\partial(\gamma\rho)}{\partial t}+\nabla\cdot(\gamma\rho\vec{v})=0 \tag{6.41}$$

$$\frac{\partial(\gamma\rho\vec{v})}{\partial t}+\nabla\cdot(\gamma\rho\vec{v}\vec{v})=-\gamma\nabla p+\nabla\cdot(\gamma\vec{\tau})+\gamma\vec{B}_f-\left(\frac{\gamma^2\mu}{K}\vec{v}+\frac{\gamma^3 C_2}{2}\rho|\vec{v}|\vec{v}\right) \tag{6.42}$$

The last term in Equation 6.42 (p. 284) represents the viscous and inertial drag forces imposed by the pore walls on the fluid.

Important

Note that even when you solve for the physical velocity in Equation 6.42 (p. 284), the two resistance coefficients can still be derived using the superficial velocity as given in Defining the Viscous and Inertial Resistance Coefficients (p. 270). ANSYS Fluent assumes that the inputs for these resistance coefficients are based upon well-established empirical correlations that are usually based on superficial velocity. Therefore, ANSYS Fluent automatically converts the inputs for the resistance coefficients into those that are compatible with the physical velocity formulation.

Important

Note that the inlet mass flow is also calculated from the superficial velocity. Therefore, for the same mass flow rate at the inlet and the same resistance coefficients, for either the physical or superficial velocity formulation you should obtain the same pressure drop across the porous media zone.

6.2.3.8.2. Multiphase Porous Media

You can simulate porous media multiphase flows using the **Physical Velocity Porous Formulation** to solve the true or physical velocity field throughout the entire flow field, including both porous and non-porous regions. In this approach, assuming a general scalar in the q^{th} phase, ϕ_q, the governing equation in an isotropic porous medium takes on the following form:

$$\frac{\partial\left(\gamma\alpha_q\rho_q\phi_q\right)}{\partial t}+\nabla\cdot\left(\gamma\alpha_q\rho_q\vec{v}_q\phi_q\right)=\nabla\cdot\left(\gamma\Gamma_q\nabla\phi_q\right)+\gamma S_{\phi,q} \tag{6.43}$$

Here γ is the porosity, which may vary with time and space; ρ_q is the phase density; α_q is the volume fraction; $\vec{v}_q$ is the phase velocity vector; $S_{\phi,q}$ is the source term; and Γ_q is the diffusion coefficient.

The general scalar equation Equation 6.43 (p. 284) applies to all other transport equations in the Eulerian multiphase model, such as the granular phase momentum and energy equations, turbulence modeling equations, and the species transport equations.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assuming isotropic porosity and multiphase flows, the governing equations in the q^{th} phase, Equation 17.167, Equation 17.168, and Equation 17.171 in the Theory Guide take the general forms described below.

6.2.3.8.2.1. The Continuity Equation

$$\frac{\partial}{\partial t}\left(\gamma\alpha_q\rho_q\right)+\nabla\cdot\left(\gamma\alpha_q\rho_q\vec{v}_q\right)=\gamma\sum_{p=1}^{n}\left(\dot{m}_{pq}-\dot{m}_{qp}\right)+\gamma S_q \tag{6.44}$$

6.2.3.8.2.2. The Momentum Equation

$$\begin{aligned}\frac{\partial}{\partial t}\left(\gamma\alpha_q\rho_q\vec{v}_q\right)+\nabla\cdot\left(\gamma\alpha_q\rho_q\vec{v}_q\vec{v}_q\right)= \quad & -\gamma\alpha_q\nabla p+\nabla\cdot\left(\gamma\bar{\bar{\tau}}_q\right)+\gamma\alpha_q\rho_q\vec{g}\\ & +\gamma\sum_{p=1}^{n}\left(\vec{R}_{pq}+\dot{m}_{pq}\vec{v}_{pq}-\dot{m}_{qp}\vec{v}_{qp}\right)\\ & +\gamma\left(\vec{F}_q+\vec{F}_{lift,q}+\vec{F}_{vm,q}\right)\\ & -\alpha_q\left(\frac{\gamma^2\mu}{K_q}\vec{v}_q+\frac{\gamma^3 C_{2,q}}{2}\rho_q|\vec{v}_q|\vec{v}_q\right)\end{aligned} \tag{6.45}$$

where the last term in Equation 6.45 (p. 285) is the momentum resistance (sink) source in a porous medium. It consists of two parts: a viscous loss term, and an inertial loss term. The parameter K is the permeability, and C_2 is the inertial resistance factor. Both K and C_2 are functions of $(1-\gamma)$. When $\gamma=1$, the flow is non-porous and the two loss terms disappear. Details about the user inputs related to the momentum resistance sources can be found in User Inputs for Porous Media (p. 267).

6.2.3.8.2.3. The Energy Equation

$$\begin{aligned}\frac{\partial}{\partial t}\left(\gamma\alpha_q\rho_q h_q\right)+\nabla\cdot\left(\gamma\alpha_q\rho_q\vec{v}_q h_q\right)= \quad & -\gamma\alpha_q\frac{\partial p_q}{\partial t}+\gamma\bar{\bar{\tau}}_q:\nabla\vec{v}_q-\nabla\cdot\left(\gamma\vec{q}_q\right)+\gamma S_q\\ & +\gamma\sum_{p=1}^{n}\left(Q_{pq}+\dot{m}_{pq}h_{pq}-\dot{m}_{qp}h_{qp}\right)+Q_{sp}\end{aligned} \tag{6.46}$$

where Q_{sp} is the heat transfer between the solids surface and the phase q in a porous medium. Assuming only convective heat transfer, we then have

$$Q_{sp}=\left(1-\gamma\right)\alpha_q h_{q,eff}\left(T_s-T_q\right) \tag{6.47}$$

where $h_{q,eff}$ is the effective convective heat transfer coefficient, and T_s is the solids surface temperature in the porous medium. It is governed by the heat conduction equation:

$$\frac{\partial}{\partial t}\left(\rho_s h_s\right)+\nabla\cdot\left(\vec{v}_s\rho_s h_s\right)=\nabla\cdot\left(k_s\nabla T\right)-\sum_{p=1}^{n}Q_{sp} \tag{6.48}$$

Equation 6.48 (p. 285) can be solved as a user-defined scalar (UDS) equation, as described in User-Defined Scalar (UDS) Transport Equations (p. 573). By default, ANSYS Fluent assumes that the overall heat transfer between the multiphase fluid and the solids is in equilibrium. Therefore, instead of solving Equation 6.48 (p. 285) to obtain the solids surface temperature, we have

$$\sum_{p=1}^{n}Q_{sp}=0 \tag{6.49}$$

and then

$$T_s = \frac{\sum_{p=1}^{n} \alpha_p h_{p,eff} T_p}{\sum_{p=1}^{n} \alpha_p h_{p,eff}} \tag{6.50}$$

6.2.3.9. Solution Strategies for Porous Media

In general, you can use the standard solution procedures and solution parameter settings when your ANSYS Fluent model includes porous media. You may find, however, that the rate of convergence slows when you define a porous region through which the pressure drop is relatively large in the flow direction (for example, the permeability, α, is low or the inertial factor, C_2, is large). This slow convergence can occur because the porous media pressure drop appears as a momentum source term—yielding a loss of diagonal dominance—in the matrix of equations solved. The best remedy for poor convergence of a problem involving a porous medium is to supply a good initial guess for the pressure drop across the medium.

Similarly, **Standard Initialization** is the recommended initialization method for porous media simulations. The default **Hybrid Initialization** method does not account for the porous media properties, and depending on boundary conditions, may produce an unrealistic initial velocity field. For porous media simulations, the **Hybrid Initialization** method should only be used with the **Maintain Constant Velocity Magnitude** option.

Another possible way to deal with poor convergence is to temporarily disable the porous media model (by turning off the **Porous Zone** option in the Fluid Dialog Box (p. 2369)) and obtain an initial flow field without the effect of the porous region. With the porous media model turned off, ANSYS Fluent will treat the porous zone as a fluid zone and calculate the flow field accordingly. Once an initial solution is obtained, or the calculation is proceeding steadily to convergence, you can enable the porous media model and continue the calculation with the porous region included. (This method is not recommended for porous media with high resistance.)

Simulations involving highly anisotropic porous media may, at times, pose convergence troubles. You can address these issues by limiting the anisotropy of the porous media coefficients ($1/\ \alpha_{ij}$ and $C_{2_{i,j}}$) to two or three orders of magnitude. Even if the medium's resistance in one direction is infinite, you do not need to set the resistance in that direction to be greater than 1000 times the resistance in the primary flow direction.

6.2.3.10. Postprocessing for Porous Media

The impact of a porous region on the flow field can be determined by examining either velocity components or pressure values. Graphical plots (including XY plots and contour or vector plots) or alphanumeric reports of the following variables/functions may be of interest:

- **X**, **Y**, **Z Velocity** (in the **Velocity...** category)
- **Static Pressure** (in the **Pressure...** category)

These variables are contained in the specified categories of the variable selection drop-down list that appears in postprocessing dialog boxes.

Note that thermal reporting in the porous region is defined as follows:

$$k_{eff} = \gamma k_f + \left(1-\gamma\right) k_s \tag{6.51}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where

γ = porosity of the medium

k_f = fluid phase thermal conductivity (including the turbulent contribution, k_t)

k_s = solid medium thermal conductivity

Important

For porous media involving surface reactions, you can display/report the surface reaction rates using the **Arrhenius Rate of Reaction-n** in the **Reactions...** category of the variable selection drop-down list.

Important

Special care must be taken when postprocessing a porous media simulation that utilizes the non-equilibrium thermal model. See Non-Equilibrium Thermal Model (p. 277) for details.

6.2.4. 3D Fan Zones

3D fan zones are fluid cell zones that simulate the effect of an axial fan by applying a distributed momentum source in a toroid-shaped fluid volume (that is, a blade-swept volume). This is in contrast to fan boundary conditions (for example, a boundary zone of type **fan** or **intake-fan**), which are based on a lumped parameter model in which the fan is considered to be infinitely thin and a discontinuous pressure rise is specified as a function of velocity. 3D fan zones have the following advantages:

- 3D fan zones can calculate swirl and radial velocities, as opposed to fan boundary conditions, which are mainly used to simulate the axial flow and require you to input swirl and radial coefficients.
- The results for 3D fan zones can be comparable to moving reference frame (MRF) simulations, and do not require modeling 3D rotating fan blade geometries.
- 3D fan zones can simulate a fan even if the available fan curve data was measured at different operating conditions than that of the simulation.

Note

Fluid zones designated as 3D fan zones cannot have non-conformal interfaces.

6.2.4.1. Momentum Equations for 3D Fan Zones

The following equations are used in the 3D fan zone for the momentum sources in the axial, tangential, and radial directions, which mimic the effect of the fan on the fluid. The tangential momentum source is based on a turbomachinery relation, whereas the radial momentum source is based on a centrifugal force balance.

$$\text{Axial: } S_{\text{a}} = \Delta P(Q)/h \tag{6.52}$$

$$\text{Tangential: } S_{\text{t}} = \begin{cases} \dfrac{2W_{\text{fan}} r}{c_1 |\Omega_{\text{operating}}|} & \text{for } R_{\text{h}} < r \le R_{\text{ip}} \\ \dfrac{2W_{\text{fan}} R_{\text{ip}}^2}{c_1 r |\Omega_{\text{operating}}|} & \text{for } R_{\text{ip}} < r < R_{\text{t}} \end{cases} \tag{6.53}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\text{Radial: } S_{\text{r}} = \frac{\rho V_{\phi}^2}{r} \tag{6.54}$$

where

$\Delta P(Q)$ = pressure rise across the fan for a given axial flow rate Q

h = thickness of the toroid region swept by the blades in the axial direction

W_{fan} = fan power

r = local radial distance from the fan axis

$\Omega_{\text{operating}}$ = fan operating angular velocity

R_{h} = radius of the fan hub

R_{ip} = radius of a point on the fan blade based on the inflection point ratio

R_{t} = radius of the fan blade tip

ρ = fluid density

V_{ϕ} = local tangential velocity

In order to calculate the tangential source, three other expressions are necessary:

$$W_{\text{fan}} = \left(\mathbf{v}_{\text{exit}} \cdot \mathbf{a}_{\text{exit}}\right) \Delta P(Q) \tag{6.55}$$

$$R_{\text{ip}} = R_{\text{h}} + \left(R_{\text{t}} - R_{\text{h}}\right) b_{\text{ip}} \tag{6.56}$$

$$c_1 = \pi h \left(R_{\text{ip}}^2 \left(2R_{\text{t}}^2 - R_{\text{ip}}^2\right) - R_{\text{h}}^4\right) \tag{6.57}$$

where

$\mathbf{v}_{\text{exit}}$ = velocity vector at the fan exit plane

$\mathbf{a}_{\text{exit}}$ = area vector at the fan exit plane

b_{ip} = the inflection point ratio, that is, the fraction of the fan blade length (starting at the hub) over which the tangential velocity of the fan discharge is increasing with increasing radius and peaks, before tapering off (see Figure 6.17: The Inflection Point Ratio of a Pitched Blade Turbine (p. 291))

The pressure rise across the 3D fan zone is specified as a function of the volumetric flow rate through the fan. The relationship may be a constant or a fan curve, that is, a polynomial or piecewise-linear function. In the case of a polynomial, the relationship is of the form

$$\Delta P(Q) = \sum_{n=1}^{N} f_n Q^{n-1} \tag{6.58}$$

where f_n are the pressure head polynomial coefficients (calculated by Fluent), and Q is the flow rate normal to the fan.

Important

The flow rate Q can be either positive or negative. You must be careful to model the fan so that a pressure rise occurs for forward flow through the fan.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The fan curve data is typically collected at a prescribed set of test conditions. However, the operating conditions of your CFD simulation may be inconsistent with this data (for example, the rotational speed or fluid temperature may be different). To account for this, the 3D fan model can adjust the fan curve data according to the following scaling relationships:

$$P_{\text{operating}}(Q) = P_{\text{test}}(Q)\sqrt{\frac{\Omega_{\text{operating}}}{\Omega_{\text{test}}}} \tag{6.59}$$

$$Q_{\text{operating}} = Q_{\text{test}}\sqrt{\frac{\Omega_{\text{operating}}}{\Omega_{\text{test}}}} \tag{6.60}$$

$$W_{\text{fan,operating}} = W_{\text{fan,test}}\frac{T_{\text{test}}}{T_{\text{avg}}} \tag{6.61}$$

where T_{test} and T_{avg} are the test temperature and the volume-averaged temperature of the fan cell zone (which is calculated internally), respectively.

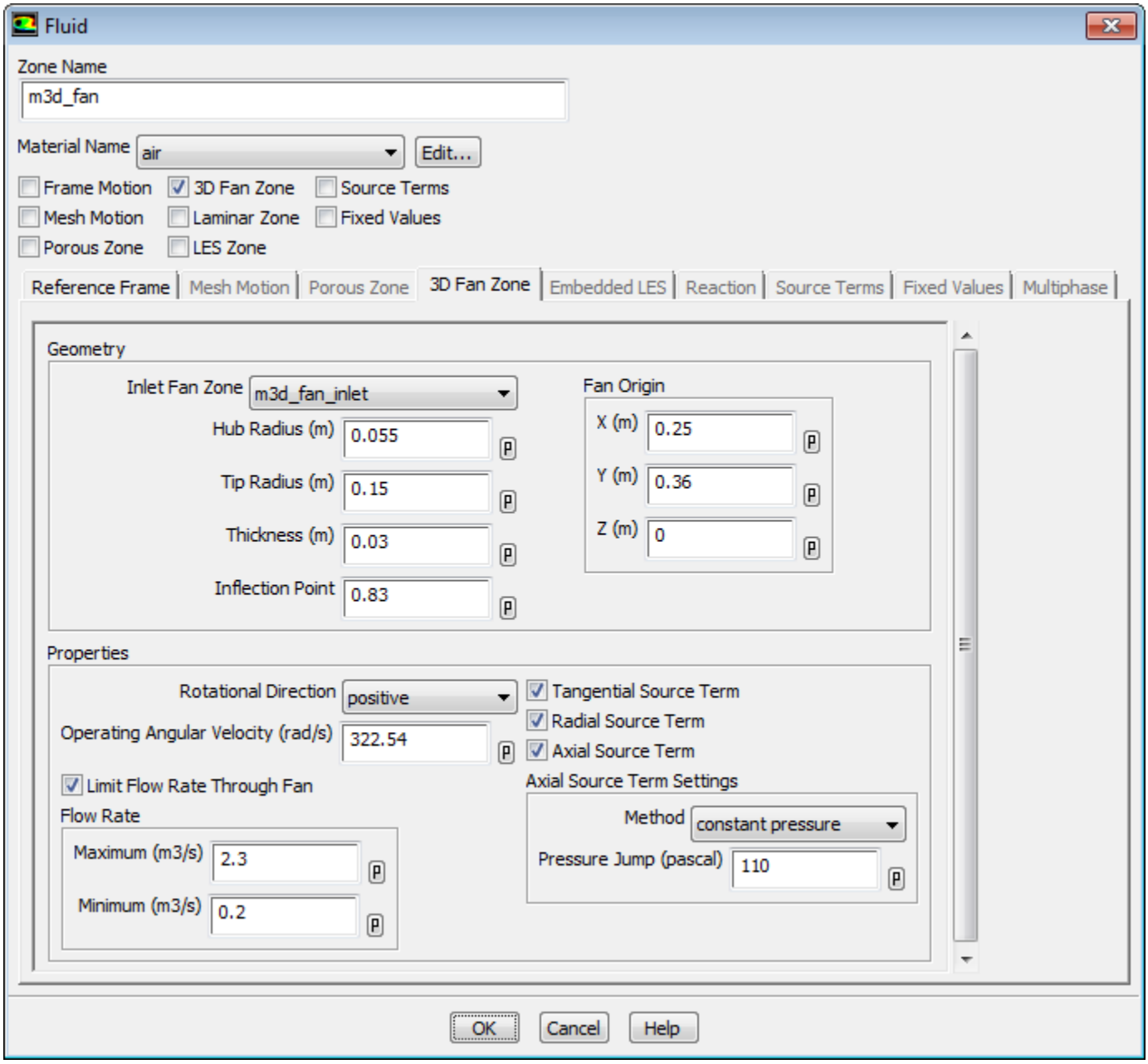
6.2.4.2. User Inputs for 3D Fan Zones

A 3D fan zone is modeled as a special type of fluid zone. The cell zone should have a toroid shape that is sized to match the blade-swept volume of the fan you are simulating (that is, has an inner and outer radius that matches the radius of the fan's hub and blade tips, respectively, as well as a length that matches the thickness of the toroid region swept by the blades in the axial direction). Furthermore, the fluid cell zone should be "interior", that is, have at least two boundary zones of type **interior** that border another fluid cell zone and act as the fan inlet and outlet (the other boundaries can be of type **interior** as well).

To designate a cell zone as a 3D fan zone, enable the **3D Fan Zone** option in the **Fluid** dialog box and then define the geometry and properties of the fan in the **3D Fan Zone** tab.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.16: The Fluid Dialog Box for a 3D Fan Zone



6.2.4.2.1. Defining the Geometry of a 3D Fan Zone

After enabling the **3D Fan Zone** option in the **Fluid** dialog box, you can begin setting it up by defining the settings in the **Geometry** group box in the **3D Fan Zone** tab of the fluid cell zone:

1. Select a boundary zone from the **Inlet Fan Zone** drop-down list, which will act as the inlet zone for the 3D fan zone and the surface on which the fan will be located. This boundary must meet certain requirements: it must be of type **interior**; it must be planar; and it must act as a boundary between the 3D fan zone and another fluid cell zone. Note that the direction the fan blows will be *into* the 3D fan zone, perpendicular to this boundary zone.

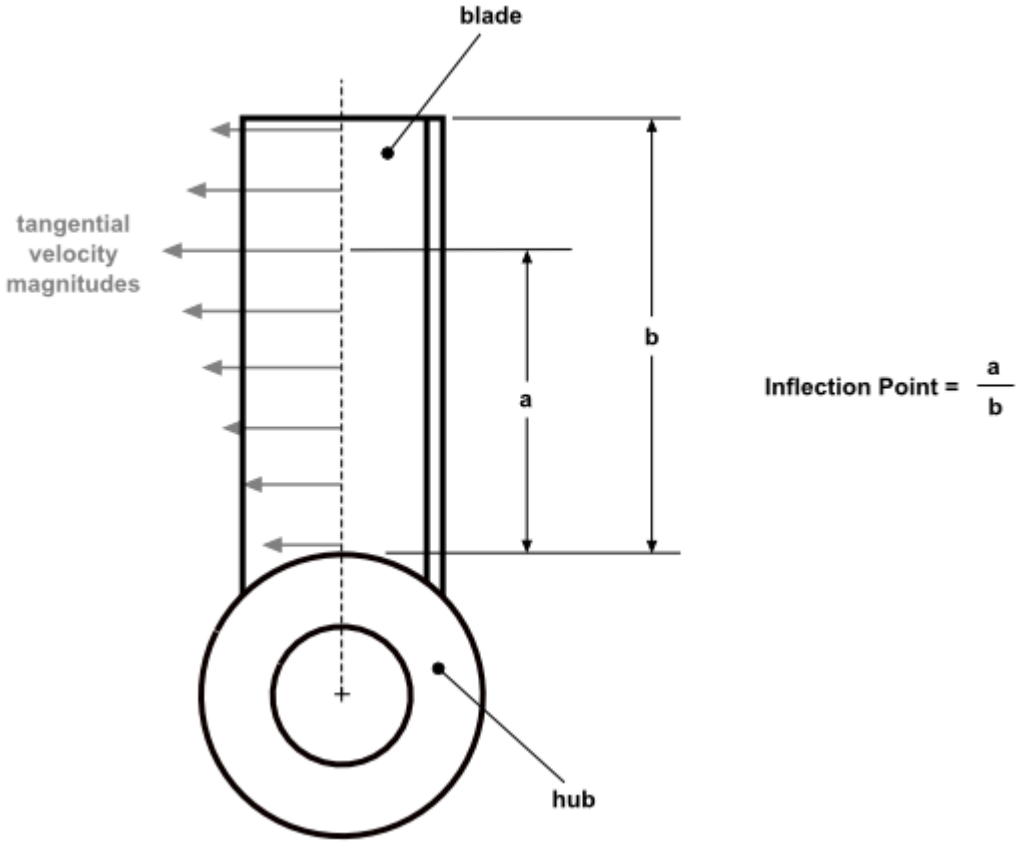
2. Define the overall dimensions of the fan by entering values for the **Hub Radius** and the **Tip Radius**, as well as the **Thickness** of the toroid region swept by the blades in the axial direction. Note that the values

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

must fit within the confines of the 3D fan zone, and that the **Thickness** will be applied from the **Inlet Fan Zone** into the 3D fan zone.

3. Specify the **Inflection Point** ratio. The inflection point ratio represents the fraction of the fan blade length (starting at the hub) over which the tangential velocity of the fan discharge is increasing with increasing radius and peaks, before tapering off. See Figure 6.17: The Inflection Point Ratio of a Pitched Blade Turbine (p. 291) for an illustration. You can derive this ratio empirically from experimental data; otherwise, you can first use ANSYS Fluent to model the actual fan geometry (for example, using a moving reference frame simulation) and then use those results to determine the input parameters for simpler simulations using the **3D Fan Zone** option.

Figure 6.17: The Inflection Point Ratio of a Pitched Blade Turbine



4. Enter the coordinates for the **Fan Origin**. This must define a point that is on the plane of the **Inlet Fan Zone** and located such that the diameters of the fan fall within boundaries of the **Inlet Fan Zone**.

6.2.4.2.2. Defining the Properties of a 3D Fan Zone

To complete the setup of a 3D fan zone, you must define the settings in the **Properties** group box in the **3D Fan Zone** tab of the fluid cell zone:

1. Define the **Rotational Direction** of the fan. The rotation is deemed **positive** if it follows the right-hand rule with respect to the fan direction vector (that is, a vector from the **Fan Origin** that points into the 3D fan zone perpendicular to the **Inlet Fan Zone**).

2. Specify the **Operating Angular Velocity** of the fan. This must be a positive nonzero number.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. You can enable the **Limit Flow Rate Through Fan** option, and thereby specify the **Maximum** and **Minimum** flow rates that are allowed.

4. You can enable the **Tangential Source Term**, **Radial Source Term**, and/or **Axial Source Term** options, in order to enable momentum sources in the specified directions. Note that the tangential and radial sources are defined according to impeller theory, and so no further inputs are necessary, whereas the axial source term requires additional input (as described in the step that follows).

5. If you have enabled the **Axial Source Term** option, you must define this source in the **Axial Source Term Settings** group box. For the **Method**, you can select one of the following choices:

   - **constant pressure**

     With this method, you must then specify a constant **Pressure Jump** that is applied to all of the cells in the 3D fan zone.

   - **fan curve**

     This method allows you to specify the pressure jump through a fan curve (pressure rise vs. volumetric flow rate). Using the **Read Fan Curve...** button, you can read in a tab-delimited ASCII file that defines the relationship between pressure and flow rate for the 3D fan cell zone (see 3D Fan Curve File Format (p. 2869) for details). You must then specify the **Curve Fitting Method** Fluent should use on the data points in your file, selecting either **polynomial** or **piecewise-linear** (for the former, you must then define the **Order of Polynomial**). Finally, you must provide an estimate of the **Initial Flow Rate** through your fan zone, as well as the **Test Angular Velocity** and **Test Temperature** of the experiment that produced the data points.

     Note that a single file can define multiple cell zones, but the file must be read in each of the applicable **Fluid** dialog boxes.

6.2.4.3. 3D Fan Zone Limitations

Note the following limitations when setting up a 3D fan zone:

- 3D fan zones are not supported with the premixed combustion model.
- 3D fan zones are not supported with the Eulerian multiphase model.
- The **Energy Equation** must be enabled in the Energy Dialog Box (p. 2175) for 3D fan zones when **fan curve** is selected for the axial source term method.

6.2.5. Fixing the Values of Variables

The option to fix values of variables in ANSYS Fluent allows you to set the value of one or more variables in a fluid or solid zone, essentially setting a boundary condition for the variables within the cells of the zone. When a variable is fixed in a given cell, the transport equation for that variable is not solved in the cell (and the cell is not included when the residual sum is computed for that variable). The fixed

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

value is used for the calculation of face fluxes between the cell and its neighbors. The result is a smooth transition between the fixed value of a variable and the values at the neighboring cells.

Important

You can fix values for temperature and species mass fractions only if you are using the pressure-based solver. You can fix values for velocity components only if you are using the pressure-based segregated solver. (Refer to Pressure-Based Solver in the Theory Guide for information about the pressure-based segregated solver.)

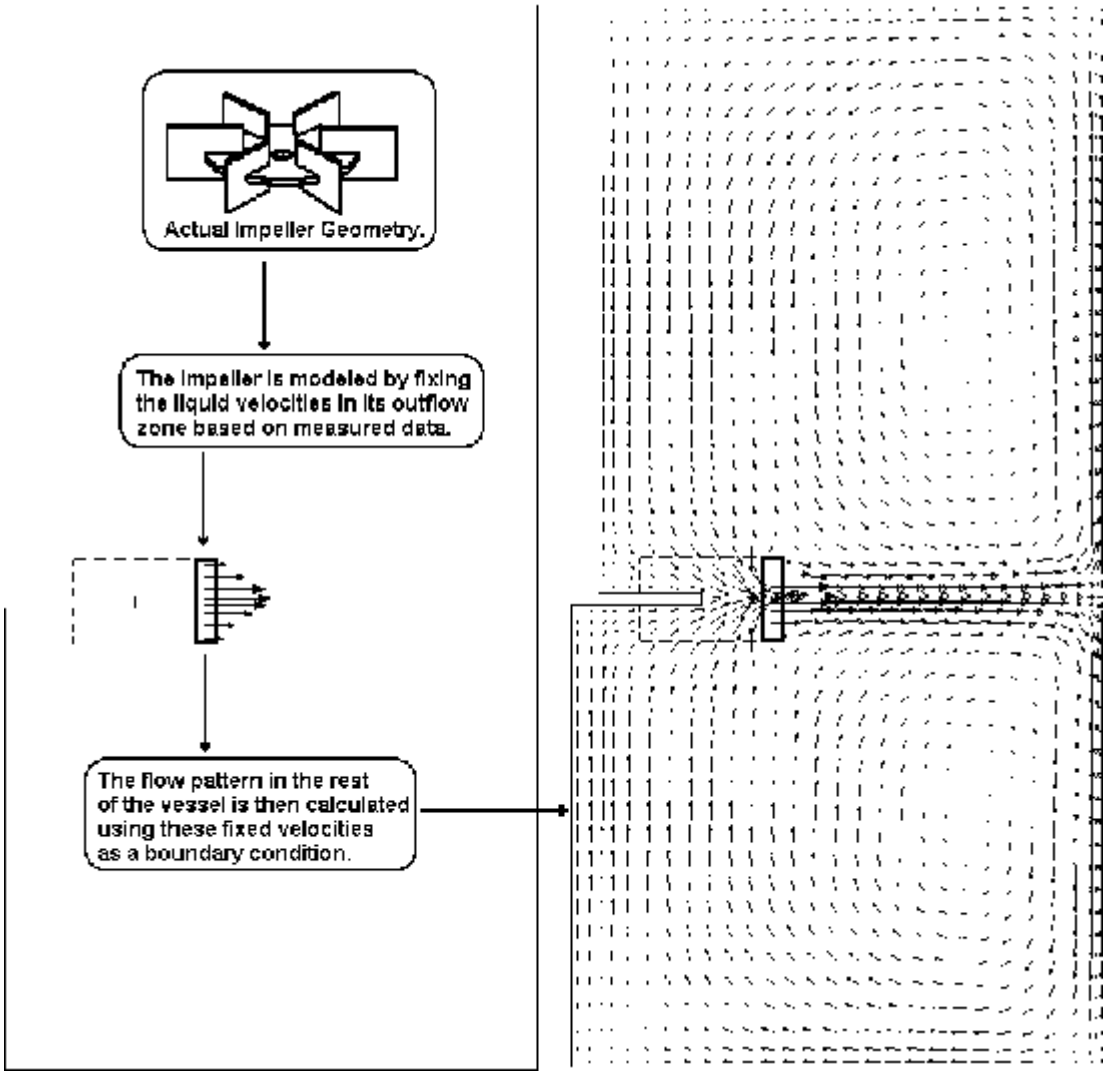
6.2.5.1. Overview of Fixing the Value of a Variable

The ability to fix the value of a variable has a wide range of applications. The velocity fixing method is often used to model the flow in stirred tanks. This approach provides an alternative to the use of a moving reference frame (solution in the reference frame of the blade) and can be used to model baffled tanks. In both 2D and 3D geometries, a fluid cell zone may be used in the impeller regions, and velocity components can be fixed based on measured data.

Although the actual impeller geometry can be modeled and the flow pattern calculated using the sliding mesh model, experimental data for the velocity profile in the outflow region are available for many impeller types. If you do not need to know the details of the flow around the blades for your problem, you can model the impeller by fixing the experimentally-obtained liquid velocities in its outflow zone. The velocities in the rest of the vessel can then be calculated using this fixed velocity profile as a boundary condition. Figure 6.18: Fixing Values for the Flow in a Stirred Tank (p. 294) shows an example of how this method is used to model the flow pattern created by a disk-turbine in an axisymmetric stirred vessel.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.18: Fixing Values for the Flow in a Stirred Tank



6.2.5.1.1. Variables That Can Be Fixed

The variables that can be fixed include velocity components (pressure-based segregated solver only), turbulence quantities, temperature (pressure-based solver only), enthalpy, species mass fractions (pressure-based solver only), and user-defined scalars. For turbulence quantities, different values can be set depending on your choice of turbulence model. You can fix the value of the temperature in a fluid or solid zone if you are solving the energy equation. If you are using the non-premixed combustion model, you can fix the enthalpy in a fluid zone. If you have more than one species in your model, you can specify fixed values for the species mass fractions for each individual species except the last one you defined. See the Fluent Customization Manual for details about defining user-defined scalars.

If you are using the Eulerian multiphase model, you can fix the values of velocity components and (depending on which multiphase turbulence model you are using) turbulence quantities on a per-phase basis. See Eulerian Model (p. 1401) for details about setting boundary conditions for Eulerian multiphase calculations.

6.2.5.2. Procedure for Fixing Values of Variables in a Zone

To fix the values of one or more variables in a cell zone, follow these steps (remembering to use only SI units):

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the Fluid Dialog Box (p. 2369) or Solid Dialog Box (p. 2379), enable the **Fixed Values** option.

2. Fix the values for the appropriate variables in the **Fixed Values** tab, noting the comments below.

   - To specify a constant value for a variable, choose **constant** in the drop-down list next to the relevant field and then enter the constant value in the field.

   - To specify a non-constant value for a variable, you can use a profile (see Profiles (p. 435)) or a user-defined function for a profile (see the Fluent Customization Manual). Select the appropriate profile or UDF in the drop-down list next to the relevant field.

     If you specify a radial-type profile (see Profile Specification Types (p. 435)) for temperature, enthalpy, species mass/mole fractions, or turbulence quantities for the k- ε, Spalart-Allmaras, or k-ω model, the local coordinate system upon which the radial profile is based is defined by the **Rotation-Axis Origin** and **Rotation-Axis Direction** for the fluid zone. See Specifying the Rotation Axis (p. 255) for information about setting these parameters. (Note that it is acceptable to specify the rotation axis and direction for a non-rotating zone. This will not cause the zone to rotate; it will not rotate unless it has been explicitly defined as a moving zone.)

   - If you do not want to fix the value for a variable, choose (or keep) **none** in the drop-down list next to the relevant field. This is the default for all variables.

6.2.5.2.1. Fixing Velocity Components

To fix the velocity components, you can specify **X**, **Y**, and (in 3D) **Z Velocity** values, or, for axisymmetric cases, **Axial**, **Radial**, and (for axisymmetric swirl) **Swirl Velocity** values. The units for a fixed velocity are m/s.

For 3D cases, you can choose to specify cylindrical velocity components instead of Cartesian components. Turn on the **Local Coordinate System For Fixed Velocities** option, and then specify the **Axial**, **Radial**, and/or **Tangential Velocity** values. The local coordinate system is defined by the **Rotation-Axis Origin** and **Rotation-Axis Direction** for the fluid zone. See Specifying the Rotation Axis (p. 255) for information about setting these parameters. (Note that it is acceptable to specify the rotation axis and direction for a non-rotating zone. This will not cause the zone to rotate; it will not rotate unless it has been explicitly defined as a moving zone.)

Important

You can fix values for velocity components only if you are using the pressure-based segregated solver. (Refer to Pressure-Based Solver in the Theory Guide for information about the pressure-based segregated solver.)

6.2.5.2.2. Fixing Temperature and Enthalpy

If you are solving the energy equation, you can fix the temperature in a zone by specifying the value of the **Temperature**. The units for a fixed temperature are K.

If you are using the non-premixed combustion model, you can fix the enthalpy in a zone by specifying the value of the **Enthalpy**. The units for a fixed enthalpy are J/kg.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you specify a radial-type profile (see Profile Specification Types (p. 435)) for temperature or enthalpy, the local coordinate system upon which the radial profile is based is defined by the **Rotation-Axis Origin** and **Rotation-Axis Direction** for the fluid zone. See above for details.

Important

You can fix the value of temperature only if you are using the pressure-based solver.

6.2.5.2.3. Fixing Species Mass Fractions

If you are using the species transport model, you can fix the values of the species mass fractions for individual species. ANSYS Fluent allows you to fix the species mass fraction for each species (for example, **h2**, **o2**) except the last one you defined.

If you specify a radial-type profile (see Profile Specification Types (p. 435)) for a species mass fraction, the local coordinate system upon which the radial profile is based is defined by the **Rotation-Axis Origin** and **Rotation-Axis Direction** for the fluid zone. See above for details.

Important

You can fix values for species mass fractions only if you are using the pressure-based solver.

6.2.5.2.4. Fixing Turbulence Quantities

To fix the values of k and ε in the k- ε equations, specify the **Turbulence Kinetic Energy** and **Turbulence Dissipation Rate** values. The units for k are m^2/s^2 and those for ε are m^2/s^3.

To fix the value of the modified turbulent viscosity ($\tilde{\nu}$) for the Spalart-Allmaras model, specify the **Modified Turbulent Viscosity** value. The units for the modified turbulent viscosity are m^2/s.

To fix the values of k and ω in the k- ω equations, specify the **Turbulence Kinetic Energy** and **Specific Dissipation Rate** values. The units for k are m^2/s^2 and those for ω are 1/s.

To fix the value of k, ε, or the Reynolds stresses in the RSM transport equations, specify the **Turbulence Kinetic Energy**, **Turbulence Dissipation Rate**, **Specific Dissipation Rate**, **UU Reynolds Stress**, **VV Reynolds Stress**, **WW Reynolds Stress**, **UV Reynolds Stress**, **VW Reynolds Stress**, and/or **UW Reynolds Stress**. The units for k and the Reynolds stresses are m^2/s^2, and those for ε are m^2/s^3.

If you specify a radial-type profile (see Profile Specification Types (p. 435)) for k, ε, ω, or $\tilde{\nu}$, the local coordinate system upon which the radial profile is based is defined by the **Rotation-Axis Origin** and **Rotation-Axis Direction** for the fluid zone. See above for details. Note that you cannot specify radial-type profiles for the Reynolds stresses.

6.2.5.2.5. Fixing User-Defined Scalars

To fix the value of a user-defined scalar, specify the **User-defined scalar-n** value. (There will be one for each user-defined scalar you have defined.) The units for a user-defined scalar will be the appropriate SI units for the scalar quantity. See the Fluent Customization Manual for information on user-defined scalars.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.2.6. Locking the Temperature for Solid and Shell Zones

You can lock (or "freeze") the temperature values for all the cells in solid zones (including cells to which you have applied an energy source using a UDF) and in walls that have shell conduction enabled, so that the values do not change during further solver iterations. When the temperature is locked for a given cell, the transport equation will still be solved in the cell, and the cell will be included when the residual sum is computed.

You can lock / unlock the temperature for solid and shell zones by using the following text command:

`solve → set → lock-solid-temperature?`

Note the following about the option for locking the temperature of solid and shell zones:

- It is only available when energy is enabled and the pressure-based solver is selected.
- When this option is used, the **BCGSTAB** option from the **Stabilization Method** drop-down list of the **Advanced Solution Controls** dialog box is not supported.
- Alternative means of monitoring the convergence of energy may be used.

6.2.7. Defining Mass, Momentum, Energy, and Other Sources

You can define volumetric sources of mass (for single or multiple species), momentum, energy, turbulence, and other scalar quantities in a fluid zone, or a source of energy for a solid zone. This feature is useful when you want to input a known value for these sources. (For more complicated sources with functional dependency, you can create a user-defined function as described in the separate Fluent Customization Manual.) To add source terms to a cell or group of cells, you must place the cell(s) in a separate zone. The sources are then applied to that cell zone. Typical uses for this feature are listed below:

- A flow source that cannot be represented by an inlet, for example, due to an issue of scale. If you need to model an inlet that is smaller than a cell, you can place the cell where the tiny "inlet" is located in its own fluid zone and then define the mass, momentum, and energy sources in that cell zone. For the example shown in Figure 6.19: Defining a Source for a Tiny Inlet (p. 298), you should set a mass source of $\frac{\dot{m}}{V} = \frac{\rho_j A_j v_j}{V}$ and a momentum source of $\frac{\dot{m}v}{V} = \frac{\dot{m}v_j}{V}$, where V is the cell volume.
- Heat release due to a source (for example, fire) that is not explicitly defined in your model. For this case, you can place the cell(s) into which the heat is originally released in its own fluid zone and then define the energy source in that cell zone.
- An energy source in a solid zone, for conjugate heat transfer applications. For this case, you can place the cell(s) into which the heat is originally released in its own solid zone and then define the energy source in that cell zone.
- A species source due to a reaction that is not explicitly included in the model. In the above example of simulating a fire, you might need to define a source for a species representing smoke generation.

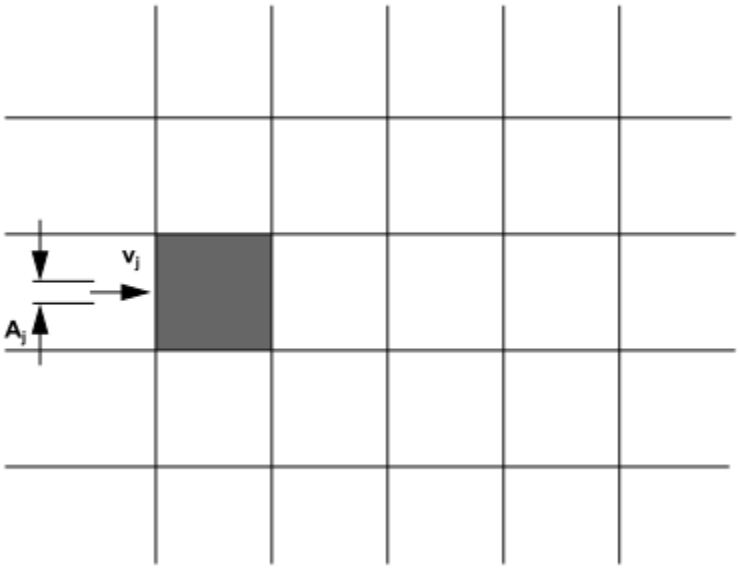
Important

Note that if you define a mass source for a cell zone, you should also define a momentum source and, if appropriate for your model, energy and turbulence sources. If you define only a mass source, that mass enters the domain with no momentum or thermal heat. The mass

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

will therefore have to be accelerated and heated by the flow and, consequently, there may be a drop in velocity or temperature. This drop may or may not be perceptible, depending on the size of the source. (Note that defining only a momentum, energy, or turbulence source is acceptable.)

Figure 6.19: Defining a Source for a Tiny Inlet



6.2.7.1. Sign Conventions and Units

All positive source terms indicate sources, and all negative source terms indicate sinks. All sources must be specified in SI units.

6.2.7.2. Procedure for Defining Sources

To define one or more source terms for a zone, follow these steps (remembering to use only SI units):

1. In the Fluid Dialog Box (p. 2369) or Solid Dialog Box (p. 2379), turn on the **Source Terms** option.
2. Set the appropriate source terms under the **Source Terms** tab, noting the comments below.
   - To specify a source, click the **Edit...** button next to the mass, momentum, energy, or other source. The sources dialog box will open where you will define the number of sources. For each source, choose **none**, **constant**, or **New Input Parameter...** in the drop-down list.
   - If you do not want to specify a source term for a variable, choose (or keep) **none** in the drop-down list next to the relevant field. This is the default for all variables.
   - To specify a constant source, choose **constant** in the drop-down list and then enter the constant value in the field.
   - To specify a new source parameter, choose **New Input Parameter...** in the drop-down list and enter the **Name** or the parameter and the **Current Value** as a constant.
   - Remember that you should not define just a mass source without defining the other sources, as described in Defining Mass, Momentum, Energy, and Other Sources (p. 297). above.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Since the sources you specify are defined per unit volume, to determine the appropriate value of your source term you will often need to first determine the volume of the cell(s) in the zone for which you are defining the source. To do this, you can use the Volume Integrals Dialog Box (p. 2620).

6.2.7.2.1. Mass Sources

If you have only one species in your problem, you can simply define a **Mass** source for that species. The units for the mass source are kg/m^3–s. In the continuity equation (Equation 1.1 in the Theory Guide), the defined mass source will appear in the S_m term.

If you have more than one species, you can specify mass sources for each individual species. There will be a total **Mass** source term as well as a source term listed explicitly for each species (for example, **h2**, **o2**) except the last one you defined. If the total of all species mass sources (including the last one) is 0, then you should specify a value of 0 for the **Mass** source, and also specify the values of the nonzero individual species mass sources. Since you cannot specify the mass source for the last species explicitly, ANSYS Fluent will compute it by subtracting the sum of all other species mass sources from the specified total **Mass** source.

For example, if the mass source for hydrogen in a hydrogen-air mixture is 0.01, the mass source for oxygen is 0.02, and the mass source for nitrogen (the last species) is 0.015, you will specify a value of 0.01 in the **h2** field, a value of 0.02 in the **o2** field, and a value of 0.045 in the **Mass** field. This concept also applies within each cell if you use user-defined functions for species mass sources.

The units for the species mass sources are kg/m^3–s. In the conservation equation for a chemical species (Equation 7.1 in the Theory Guide), the defined mass source will appear in the S_i term.

6.2.7.2.2. Momentum Sources

To define a source of momentum, specify the **X Momentum**, **Y Momentum**, and/or **Z Momentum** term. The units for the momentum source are N/m^3. In the momentum equation (Equation 1.3 in the Theory Guide), the defined momentum source will appear in the $\vec{F}$ term.

6.2.7.2.3. Energy Sources

To define a source of energy, specify an **Energy** term. The units for the energy source are W/m^3. In the energy equation (Equation 5.1 in the Theory Guide), the defined energy source will appear in the S_h term.

6.2.7.2.4. Turbulence Sources

6.2.7.2.4.1. Turbulence Sources for the k- ε Model

To define a source of k or ε in the k- ε equations, specify the **Turbulent Kinetic Energy** or **Turbulent Dissipation Rate** term. The units for the k source are kg/m-s^3 and those for ε are kg/m-s^4.

The defined k source will appear in the S_k term on the right-hand side of the turbulent kinetic energy equation (for example, Equation 4.39 in the Theory Guide).

The defined ε source will appear in the S_ε term on the right-hand side of the turbulent dissipation rate equation (for example, Equation 4.40 in the Theory Guide).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.2.7.2.4.2. Turbulence Sources for the Spalart-Allmaras Model

To define a source of modified turbulent viscosity, specify the **Modified Turbulent Viscosity** term. The units for the modified turbulent viscosity source are kg/m-s^2. In the transport equation for the Spalart-Allmaras model (Equation 4.15 in the Theory Guide), the defined modified turbulent viscosity source will appear in the $S_{\tilde{\nu}}$ term.

6.2.7.2.4.3. Turbulence Sources for the k- ω Model

To define a source of k or ω in the k- ω equations, specify the **Turbulent Kinetic Energy** or **Specific Dissipation Rate** term. The units for the k source are kg/m-s^3 and those for ω are kg/m^3-s^2.

The defined k source will appear in the S_k term on the right-hand side of the turbulent kinetic energy equation (Equation 4.70 in the Theory Guide).

The defined ω source will appear in the S_ω term on the right-hand side of the specific turbulent dissipation rate equation (Equation 4.71 in the Theory Guide).

6.2.7.2.4.4. Turbulence Sources for the Reynolds Stress Model

To define a source of k, ε, or the Reynolds stresses in the RSM transport equations, specify the **Turbulence Kinetic Energy**, **Turbulence Dissipation Rate**, **Specific Dissipation Rate**, **UU Reynolds Stress**, **VV Reynolds Stress**, **WW Reynolds Stress**, **UV Reynolds Stress**, **VW Reynolds Stress**, and/or **UW Reynolds Stress** terms. The units for the k source and the sources of Reynolds stress are kg/m-s^3, and those for ε are kg/m-s^4.

The defined Reynolds stress sources will appear in the S_{user} term on the right-hand side of the Reynolds stress transport equation (Equation 4.204 in the Theory Guide).

The defined k source will appear in the S_k term on the right-hand side of Equation 4.232 in the Theory Guide.

The defined ε will appear in the S_ε term on the right-hand side of Equation 4.235 in the Theory Guide.

6.2.7.2.5. Mean Mixture Fraction and Variance Sources

To define a source of the mean mixture fraction or its variance for the non-premixed combustion model, specify the **Mean Mixture Fraction** or **Mixture Fraction Variance** term. The units for the mean mixture fraction source are kg/m^3–s, and those for the mixture fraction variance source are kg/m^3–s.

The defined mean mixture fraction source will appear in the S_{user} term in the transport equation for the mixture fraction (Equation 8.4 in the Theory Guide).

The defined mixture fraction variance source will appear in the S_{user} term in the transport equation for the mixture fraction variance (Equation 8.5 in the Theory Guide).

If you are using the two-mixture-fraction approach, you can also specify sources of the **Secondary Mean Mixture Fraction** and **Secondary Mixture Fraction Variance**.

6.2.7.2.6. P-1 Radiation Sources

To define a source for the P-1 radiation model, specify the **P1** term. The units for the radiation source are W/m^3, and the defined source will appear in the S_G term in Equation 5.18 in the Theory Guide.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that, if the source term you are defining represents a transfer from internal energy to radiative energy (for example, absorption or emission), you must specify an **Energy** source of the same magnitude as the **P1** source, but with the opposite sign, in order to ensure overall energy conservation.

6.2.7.2.7. Progress Variable Sources

To define a source of the progress variable for the premixed combustion model, specify the **Progress Variable** term. The units for the progress variable source are kg/m^3-s, and the defined source will appear in the ρS_c term in Equation 9.1 in the Theory Guide.

6.2.7.2.8. NO, HCN, and NH3 Sources for the NOx Model

To define a source of NO, HCN, or NH_3 for the NOx model, specify the **no**, **hcn**, or **nh3** term. The units for these sources are kg/m^3-s, and the defined sources will appear in the S_{NO}, S_{HCN}, and S_{NH3} terms of Equation 14.1, Equation 14.2, and Equation 14.3 in the Theory Guide.

6.2.7.2.9. User-Defined Scalar (UDS) Sources

You can specify source term(s) for each UDS transport equation you have defined in your model. See Setting Up UDS Equations in ANSYS Fluent (p. 575) for details.

6.3. Boundary Conditions

Boundary conditions consist of external, internal, and periodic boundaries. Most of the boundary conditions are discussed in the sections that follow.

6.3.1. Flow Inlet and Exit Boundary Conditions
6.3.2. Using Flow Boundary Conditions
6.3.3. Pressure Inlet Boundary Conditions
6.3.4. Velocity Inlet Boundary Conditions
6.3.5. Mass-Flow Inlet Boundary Conditions
6.3.6. Mass-Flow Outlet Boundary Conditions
6.3.7. Inlet Vent Boundary Conditions
6.3.8. Intake Fan Boundary Conditions
6.3.9. Pressure Outlet Boundary Conditions
6.3.10. Pressure Far-Field Boundary Conditions
6.3.11. Outflow Boundary Conditions
6.3.12. Outlet Vent Boundary Conditions
6.3.13. Exhaust Fan Boundary Conditions
6.3.14. Degassing Boundary Conditions
6.3.15. Wall Boundary Conditions
6.3.16. Symmetry Boundary Conditions
6.3.17. Periodic Boundary Conditions
6.3.18. Axis Boundary Conditions
6.3.19. Fan Boundary Conditions
6.3.20. Radiator Boundary Conditions
6.3.21. Porous Jump Boundary Conditions

You can find details about the following types here:

- interface zones: Using Sliding Meshes (p. 633)
- interior zones: Interior Dialog Box (p. 2411)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- overset zones: Overset Meshes (p. 172)
- RANS/LES interface zones: Setting Up the Embedded Large Eddy Simulation (ELES) Model (p. 793)

6.3.1. Flow Inlet and Exit Boundary Conditions

ANSYS Fluent has a wide range of boundary conditions that permit flow to enter and exit the solution domain. To help you select the most appropriate boundary condition for your application, this section includes descriptions of how each type of condition is used, and what information is needed for each one. Recommendations for determining inlet values of the turbulence parameters are also provided.

6.3.2. Using Flow Boundary Conditions

This section provides an overview of flow boundaries in ANSYS Fluent and how to use them.

ANSYS Fluent provides 12 types of boundary zone types for the specification of flow inlets and exits: velocity inlet, pressure inlet, mass-flow inlet, mass-flow outlet, pressure outlet, pressure far-field, outflow, inlet vent, intake fan, outlet vent, exhaust fan, and degassing.

The inlet and exit boundary condition options in ANSYS Fluent are as follows:

- Velocity inlet boundary conditions are used to define the velocity and scalar properties of the flow at inlet boundaries.
- Pressure inlet boundary conditions are used to define the total pressure and other scalar quantities at flow inlets.
- Mass-flow inlet boundary conditions are used in compressible flows to prescribe a mass flow rate at an inlet. It is not necessary to use mass-flow inlets in incompressible flows because when density is constant, velocity inlet boundary conditions will fix the mass flow. Like pressure and velocity inlets, other inlet scalars are also prescribed.
- Pressure outlet boundary conditions are used to define the static pressure at flow outlets (and also other scalar variables, in case of backflow). The use of a pressure outlet boundary condition instead of an outflow condition often results in a better rate of convergence when backflow occurs during iteration.
- Pressure far-field boundary conditions are used to model a free-stream compressible flow at infinity, with free-stream Mach number and static conditions specified. This boundary type is available only for compressible flows.
- Outflow boundary conditions are used to model flow exits where the details of the flow velocity and pressure are not known prior to solution of the flow problem. They are appropriate where the exit flow is close to a fully developed condition, as the outflow boundary condition assumes a zero streamwise gradient for all flow variables except pressure. They are not appropriate for compressible flow calculations.
- Inlet vent boundary conditions are used to model an inlet vent with a specified loss coefficient, flow direction, and ambient (inlet) total pressure and temperature.
- Intake fan boundary conditions are used to model an external intake fan with a specified pressure jump, flow direction, and ambient (intake) total pressure and temperature.
- Outlet vent boundary conditions are used to model an outlet vent with a specified loss coefficient and ambient (discharge) static pressure and temperature.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Exhaust fan boundary conditions are used to model an external exhaust fan with a specified pressure jump and ambient (discharge) static pressure.

- Degassing boundary conditions are used to model a free surface through which dispersed gas bubbles are allowed to escape, but the continuous liquid phase is not. A typical application is a bubble column in which you want to reduce computational cost by not including the freeboard region in the simulation. The degassing boundary condition is only available for two-phase liquid-gas flows using the Eulerian multiphase model.

6.3.2.1. Determining Turbulence Parameters

When the flow enters the domain at an inlet, outlet, or far-field boundary, ANSYS Fluent requires specification of transported turbulence quantities. This section describes which quantities are needed for specific turbulence models and how they must be specified. It also provides guidelines for the most appropriate way of determining the inflow boundary values.

6.3.2.1.1. Specification of Turbulence Quantities Using Profiles

If it is important to accurately represent a boundary layer or fully-developed turbulent flow at the inlet, you should ideally set the turbulence quantities by creating a profile file (see Profiles (p. 435)) from experimental data or empirical formulas. If you have an analytical description of the profile, rather than data points, you can either use this analytical description to create a profile file, or create a user-defined function to provide the inlet boundary information. (See the Fluent Customization Manual for information on user-defined functions.)

Once you have created the profile function, you can use it as described below:

- Spalart-Allmaras model: Choose **Modified Turbulent Viscosity** or **Turbulent Viscosity Ratio** in the **Turbulence Specification Method** drop-down list and select the appropriate profile name in the drop-down list next to **Turbulent Viscosity** or **Turbulent Viscosity Ratio**. ANSYS Fluent computes the boundary value for the modified turbulent viscosity, $\tilde{\nu}$, by combining $\frac{\mu_t}{\mu}$ with the appropriate values of density and molecular viscosity.

- k- ε models: Choose **K and Epsilon** in the **Turbulence Specification Method** drop-down list and select the appropriate profile names in the drop-down lists next to **Turbulent Kinetic Energy** and **Turbulent Dissipation Rate**.

- k- ω models: Choose **K and Omega** in the **Turbulence Specification Method** drop-down list and select the appropriate profile names in the drop-down lists next to **Turbulent Kinetic Energy** and **Specific Dissipation Rate**.

- ε-based Reynolds stress models: Choose **K and Epsilon** in the **Turbulence Specification Method** drop-down list and select the appropriate profile names in the drop-down lists next to **Turbulent Kinetic Energy** and **Turbulent Dissipation Rate**. Choose **Reynolds-Stress Components** in the **Reynolds-Stress Specification Method** drop-down list and select the appropriate profile name in the drop-down list next to each of the individual Reynolds-stress components.

- ω-based Reynolds stress models: Choose **K and Omega** in the **Turbulence Specification Method** drop-down list and select the appropriate profile names in the drop-down lists next to **Specific Dissipation Rate**. Choose **Reynolds-Stress Components** in the **Reynolds-Stress Specification Method** drop-down list and select the appropriate profile name in the drop-down list next to each of the individual Reynolds-stress components.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.2.1.2. Uniform Specification of Turbulence Quantities

In some situations, it is appropriate to specify a uniform value of the turbulence quantity at the boundary where inflow occurs. Examples are fluid entering a duct, far-field boundaries, or even fully-developed duct flows where accurate profiles of turbulence quantities are unknown.

In most turbulent flows, higher levels of turbulence are generated within shear layers than enter the domain at flow boundaries, making the result of the calculation relatively insensitive to the inflow boundary values. Nevertheless, caution must be used to ensure that boundary values are not so unphysical as to contaminate your solution or impede convergence. This is particularly true of external flows where unphysically large values of effective viscosity in the free stream can "swamp" the boundary layers.

You can use the turbulence specification methods described above to enter uniform constant values instead of profiles. Alternatively, you can specify the turbulence quantities in terms of more convenient quantities such as turbulence intensity, turbulent viscosity ratio, hydraulic diameter, and turbulence length scale. The default Turbulence Specification Method is set to **Turbulent Viscosity Ratio** (for the Spalart-Allmaras model) or **Intensity and Viscosity Ratio** (for the k- ε models, the k- ω models, or the RSM). These quantities are discussed further in the following sections.

6.3.2.1.3. Turbulence Intensity

The turbulence intensity, I, is defined as the ratio of the root-mean-square of the velocity fluctuations, u', to the mean flow velocity, u_{avg}.

A turbulence intensity of 1% or less is generally considered low and turbulence intensities greater than 10% are considered high. Ideally, you will have a good estimate of the turbulence intensity at the inlet boundary from external, measured data. For example, if you are simulating a wind-tunnel experiment, the turbulence intensity in the free stream is usually available from the tunnel characteristics. In modern low-turbulence wind tunnels, the free-stream turbulence intensity may be as low as 0.05%.

For internal flows, the turbulence intensity at the inlets is totally dependent on the upstream history of the flow. If the flow upstream is under-developed and undisturbed, you can use a low turbulence intensity. If the flow is fully developed, the turbulence intensity may be as high as a few percent. The turbulence intensity at the core of a fully-developed duct flow can be estimated from the following formula derived from an empirical correlation for pipe flows:

$$I \equiv \frac{u'}{u_{avg}} = 0.16\left(Re_{D_H}\right)^{-1/8} \tag{6.62}$$

At a Reynolds number of 50,000, for example, the turbulence intensity will be 4%, according to this formula.

The default value for turbulence intensity is 5% (medium intensity).

6.3.2.1.4. Turbulence Length Scale and Hydraulic Diameter

The turbulence length scale, ℓ, is a physical quantity related to the size of the large eddies that contain the energy in turbulent flows.

In fully-developed duct flows, ℓ is restricted by the size of the duct, since the turbulent eddies cannot be larger than the duct. An approximate relationship between ℓ and the physical size of the duct is

$$\ell = \frac{0.07L}{C_\mu^{3/4}} \tag{6.63}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where L is the relevant dimension of the duct and $1/C_{\mu}^{3/4}$ ensures consistency with the definition of the turbulent length scales for one- and two-equation turbulence models. The factor of 0.07 is based on the maximum value of the mixing length in fully-developed turbulent pipe flow, where L is the diameter of the pipe. In a channel of non-circular cross-section, you can base L on the hydraulic diameter.

If the turbulence derives its characteristic length from an obstacle in the flow, such as a perforated plate, it is more appropriate to base the turbulence length scale on the characteristic length of the obstacle rather than on the duct size.

It should be noted that the relationship of Equation 6.63 (p. 304), which relates a physical dimension (L) to the turbulence length scale (ℓ), is not necessarily applicable to all situations. For most cases, however, it is a suitable approximation.

Guidelines for choosing the characteristic length L or the turbulence length scale ℓ for selected flow types are listed below:

- For fully-developed internal flows, choose the **Intensity and Hydraulic Diameter** specification method and specify the hydraulic diameter $L=D_H$ in the **Hydraulic Diameter** field.
- For flows downstream of turning vanes, perforated plates, and so on, choose the **Intensity and Length Scale** method and specify the characteristic length of the flow opening for L in the **Turbulent Length Scale** field.
- For wall-bounded flows in which the inlets involve a turbulent boundary layer, choose the **Intensity and Length Scale** method and use the boundary-layer thickness, δ_{99}, to compute the turbulence length scale, ℓ, from $\ell=0.4\delta_{99}$. Enter this value for ℓ in the **Turbulence Length Scale** field.

6.3.2.1.5. Turbulent Viscosity Ratio

The turbulent viscosity ratio, $\frac{\mu_t}{\mu}$, is directly proportional to the turbulent Reynolds number ($Re_t \equiv k^2/(\varepsilon\nu)$). Re_t is large (on the order of 100 to 1000) in high-Reynolds-number boundary layers, shear layers, and fully-developed duct flows. However, at the free-stream boundaries of most external flows, $\frac{\mu_t}{\mu}$ is fairly small. Typically, the turbulence parameters are set so that $1<\frac{\mu_t}{\mu}<10$. For internal flows values up to 100 are sensible for the turbulent viscosity ratio, $\frac{\mu_t}{\mu}$. The default value for the turbulent viscosity ratio is set to 10.

To specify quantities in terms of the turbulent viscosity ratio, you can choose **Turbulent Viscosity Ratio** (for the Spalart-Allmaras model) or **Intensity and Viscosity Ratio** (for the k-ε models, the k-ω models, or the RSM). The default value for the turbulence intensity is set to 5% (medium intensity) and the turbulent viscosity ratio, $\frac{\mu_t}{\mu}$, has a default value of 10.

6.3.2.1.6. Relationships for Deriving Turbulence Quantities

To obtain the values of transported turbulence quantities from more convenient quantities such as I, L, or $\frac{\mu_t}{\mu}$, you must typically resort to an empirical relation. Several useful relations, most of which are used within ANSYS Fluent, are presented below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.2.1.7. Estimating Modified Turbulent Viscosity from Turbulence Intensity and Length Scale

To obtain the modified turbulent viscosity, $\tilde{\nu}$, for the Spalart-Allmaras model from the turbulence intensity, I, and length scale, ℓ, the following equation can be used. Note that this equation assumes that there is no viscous damping, that is, $\tilde{\nu}=\nu_t$ (see Modeling the Turbulent Viscosity in the Theory Guide).

$$\tilde{\nu}=C_\mu\sqrt{\frac{3}{2}}\,u_{avg}I\ell \tag{6.64}$$

This formula is also used in ANSYS Fluent if you select the **Intensity and Hydraulic Diameter** specification method with the Spalart-Allmaras model. In this case, ℓ is obtained from Equation 6.63 (p. 304). C_μ ensures consistency with the definition of the eddy viscosity for k-ε and k-ω turbulence models. The value of C_μ is 0.09.

6.3.2.1.8. Estimating Turbulent Kinetic Energy from Turbulence Intensity

The relationship between the turbulent kinetic energy, k, and turbulence intensity, I, is

$$k=\frac{3}{2}\left(u_{avg}I\right)^2 \tag{6.65}$$

where u_{avg} is the mean flow velocity.

This relationship is used in ANSYS Fluent whenever the **Intensity and Hydraulic Diameter**, **Intensity and Length Scale**, or **Intensity and Viscosity Ratio** method is used instead of specifying explicit values for k and ε.

6.3.2.1.9. Estimating Turbulent Dissipation Rate from a Length Scale

If you know the turbulence length scale, ℓ, you can determine ε from the relationship

$$\varepsilon=\frac{k^{3/2}}{\ell} \tag{6.66}$$

The determination of ℓ was discussed previously.

This relationship is used in ANSYS Fluent whenever the **Intensity and Hydraulic Diameter** or **Intensity and Length Scale** method is used instead of specifying explicit values for k and ε.

6.3.2.1.10. Estimating Turbulent Dissipation Rate from Turbulent Viscosity Ratio

The value of ε can be obtained from the turbulent viscosity ratio $\frac{\mu_t}{\mu}$ and k using the following relationship:

$$\varepsilon=\rho C_\mu\frac{k^2}{\mu}\left(\frac{\mu_t}{\mu}\right)^{-1} \tag{6.67}$$

where C_μ is an empirical constant specified in the turbulence model.

This relationship is used in ANSYS Fluent whenever the **Intensity and Viscosity Ratio** method is used instead of specifying explicit values for k and ε.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.2.1.11. Estimating Turbulent Dissipation Rate for Decaying Turbulence

If you are simulating a wind-tunnel situation in which the model is mounted in the test section downstream of a mesh and/or wire mesh screens, you can choose a value of ε such that

$$\varepsilon \approx \frac{\Delta k U_\infty}{L_\infty} \tag{6.68}$$

where Δk is the approximate decay of k you want to have across the flow domain (say, 10% of the inlet value of k), U_∞ is the free-stream velocity, and L_∞ is the streamwise length of the flow domain. Equation 6.68 (p. 307) is a linear approximation to the power-law decay observed in high-Reynolds-number isotropic turbulence. Its basis is the exact equation for k in decaying turbulence, $U \partial k / \partial x = -\varepsilon$.

If you use this method to estimate ε, you should also check the resulting turbulent viscosity ratio $\frac{\mu_t}{\mu}$ to make sure that it is not too large, using Equation 6.67 (p. 306).

Although this method is not used internally by ANSYS Fluent, you can use it to derive a constant free-stream value of ε that you can then specify directly by choosing **K and Epsilon** in the **Turbulence Specification Method** drop-down list. In this situation, you will typically determine k from I using Equation 6.65 (p. 306).

6.3.2.1.12. Estimating Specific Dissipation Rate from a Length Scale

If you know the turbulence length scale, ℓ, you can determine ω from the relationship

$$\omega = \frac{k^{1/2}}{C_\mu \ell} \tag{6.69}$$

where C_μ is an empirical constant specified in the turbulence model. The determination of ℓ was discussed previously.

This relationship is used in ANSYS Fluent whenever the **Intensity and Hydraulic Diameter** or **Intensity and Length Scale** method is used instead of specifying explicit values for k and ω.

6.3.2.1.13. Estimating Specific Dissipation Rate from Turbulent Viscosity Ratio

The value of ω can be obtained from the turbulent viscosity ratio $\frac{\mu_t}{\mu}$ and k using the following relationship:

$$\omega = \rho \frac{k}{\mu} \left(\frac{\mu_t}{\mu} \right)^{-1} \tag{6.70}$$

This relationship is used in ANSYS Fluent whenever the **Intensity and Viscosity Ratio** method is used instead of specifying explicit values for k and ω.

6.3.2.1.14. Estimating Reynolds Stress Components from Turbulent Kinetic Energy

When the RSM is used, if you do not specify the values of the Reynolds stresses explicitly at the inlet using the **Reynolds-Stress Components** option in the **Reynolds-Stress Specification Method** drop-down list, they are approximately determined from the specified values of k. The turbulence is assumed to be isotropic such that

$$\overline{u_i' u_j'} = 0 \tag{6.71}$$

and

$$\overline{u'_\alpha u'_\alpha} = \frac{2}{3}k \tag{6.72}$$

(no summation over the index α).

ANSYS Fluent will use this method if you select **K or Turbulence Intensity** in the **Reynolds-Stress Specification Method** drop-down list.

6.3.2.1.15. Specifying Inlet Turbulence for LES

The turbulence intensity value specified at a velocity inlet for LES, as described in Large Eddy Simulation Model (p. 810), is used to randomly perturb the instantaneous velocity field at the inlet. It does not specify a modeled turbulence quantity. Instead, the stochastic components of the flow at the inlet boundary are accounted for by superposing random perturbations on individual velocity components as described in Inlet Boundary Conditions for the LES Model in the Theory Guide.

6.3.3. Pressure Inlet Boundary Conditions

Pressure inlet boundary conditions are used to define the fluid pressure at flow inlets, along with all other scalar properties of the flow. They are suitable for both incompressible and compressible flow calculations. Pressure inlet boundary conditions can be used when the inlet pressure is known but the flow rate and/or velocity is not known. This situation may arise in many practical situations, including buoyancy-driven flows. Pressure inlet boundary conditions can also be used to define a "free" boundary in an external or unconfined flow.

For an overview of flow boundaries, see Flow Inlet and Exit Boundary Conditions (p. 302).

6.3.3.1. Inputs at Pressure Inlet Boundaries

6.3.3.1.1. Summary

You will enter the following information for a pressure inlet boundary:

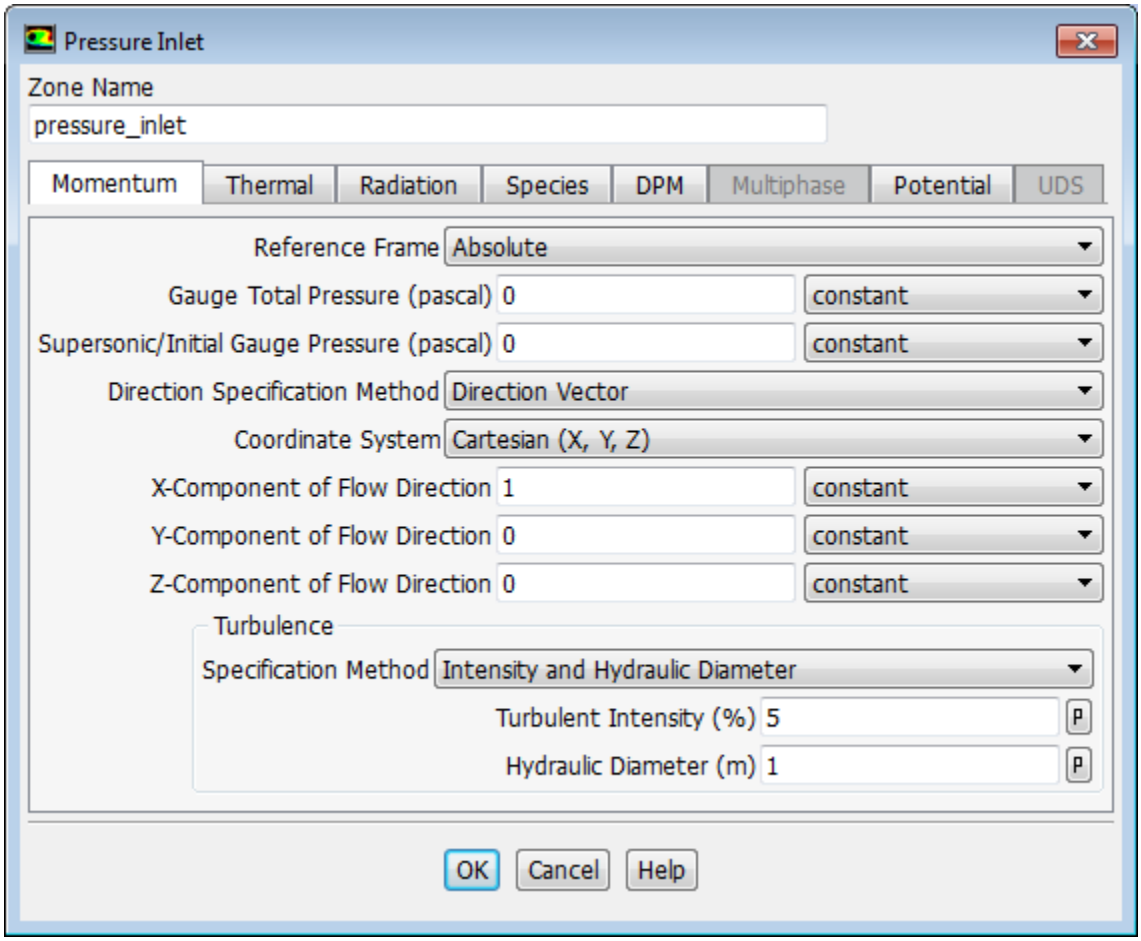
- type of reference frame
- total (stagnation) pressure
- total (stagnation) temperature
- flow direction
- static pressure
- turbulence parameters (for turbulent calculations)
- radiation parameters (for calculations using the P-1, DTRM, DO, surface-to-surface, or MC models)
- chemical species mass or mole fractions (for species calculations)
- mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
- progress variable (for premixed or partially premixed combustion calculations)
- discrete phase boundary conditions (for discrete phase calculations)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- multiphase boundary conditions (for general multiphase calculations)
- open channel flow parameters (for open channel flow calculations using the VOF multiphase model)
- acoustic wave model settings

All values are entered in the Pressure Inlet Dialog Box (p. 2434) (Figure 6.20: The Pressure Inlet Dialog Box (p. 309)), which is opened from the **Boundary Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 241)). Note that open channel boundary condition inputs are described in Modeling Open Channel Flows (p. 1410), and acoustic wave model settings are described in Boundary Acoustic Wave Models (p. 405).

Figure 6.20: The Pressure Inlet Dialog Box



6.3.3.1.1.1. Pressure Inputs and Hydrostatic Head

When gravitational acceleration is activated in the **Operating Conditions** dialog box (accessed from the **Boundary Conditions** task page), the pressure field (including all pressure inputs) will include the hydrostatic head. This is accomplished by redefining the pressure in terms of a modified pressure that includes the hydrostatic head (denoted p') as follows:

$$p' = p - \rho_0 \vec{g} \cdot \vec{r} \tag{6.73}$$

where ρ_0 is a constant reference density, $\vec{g}$ is the gravity vector (also a constant), and

$$\vec{r} = x\hat{i} + y\hat{j} + z\hat{k} \tag{6.74}$$

is the position vector. Noting that

$$\nabla\left(\rho_0\vec{g}\cdot\vec{r}\right)=\rho_0\vec{g} \tag{6.75}$$

it follows that

$$\nabla p'=\nabla\left(p-\rho_0\vec{g}\cdot\vec{r}\right)=\nabla p-\rho_0\vec{g} \tag{6.76}$$

The substitution of this relation in the momentum equation gives pressure gradient and gravitational body force terms of the form

$$-\nabla p'+\left(\rho-\rho_0\right)\vec{g} \tag{6.77}$$

where ρ is the fluid density. Therefore, if the fluid density is constant, we can set the reference density ρ_0 equal to the fluid density, thereby eliminating the body force term. If the fluid density is not constant (for example, density is given by the ideal gas law), then the reference density should be chosen to be representative of the average or mean density in the fluid domain, so that the body force term is small.

An important consequence of this treatment of the gravitational body force is that your inputs of pressure (now defined as p') should not include hydrostatic pressure differences. Moreover, reports of static and total pressure will not show any influence of the hydrostatic pressure. See Natural Convection and Buoyancy-Driven Flows (p. 833) for additional information.

6.3.3.1.1.2. Defining Total Pressure and Temperature

Enter the value for total pressure in the **Gauge Total Pressure** field in the **Pressure Inlet** dialog box. Total temperature is set in the **Thermal** tab, in the **Total Temperature** field.

Remember that the total pressure value is the gauge pressure with respect to the operating pressure defined in the Operating Conditions Dialog Box (p. 2382). Total pressure for an incompressible fluid is defined as

$$p_0=p_s+\frac{1}{2}\rho|\vec{v}|^2 \tag{6.78}$$

and for a compressible fluid of constant c_p as

$$p_0=p_s\left(1+\frac{\gamma-1}{2}M^2\right)^{\gamma/(\gamma-1)} \tag{6.79}$$

where

p_0 = total pressure

p_s = static pressure

M = Mach number

γ = ratio of specific heats $\left(c_p/\ c_v\right)$

If you are modeling axisymmetric swirl, $\vec{v}$ in Equation 6.78 (p. 310) will include the swirl component.

The **Total Temperature**, **Gauge Total Pressure**, and flow directions are in absolute or relative to the adjacent cell zone reference frames, based on the **Reference Frame** setting in the **Pressure Inlet** dialog box.

If the cell zone adjacent to a pressure inlet is defined as a moving reference frame zone, and you are using the pressure-based solver, the velocity in Equation 6.78 (p. 310) (or the Mach number in Equation 6.79 (p. 310)) will be absolute or relative to the mesh velocity, depending on whether or not the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Absolute velocity formulation is enabled in the **General** task page. For the density-based solver, the **Absolute** velocity formulation is always used; hence, the velocity in Equation 6.78 (p. 310) (or the Mach number in Equation 6.79 (p. 310)) is always the **Absolute** velocity.

For the Eulerian multiphase model, the total temperature, and velocity components must be specified for the individual phases. The **Reference Frame** (**Relative to Adjacent Cell Zone** or **Absolute**) for each of the phases is the same as the reference frame selected for the mixture phase. Note that the total pressure values must be specified in the mixture phase.

Important

- If the flow is incompressible, then the temperature assigned in the **Pressure Inlet** dialog box will be considered the static temperature.
- For the mixture multiphase model, if a boundary allows a combination of compressible and incompressible phases to enter the domain, then the temperature assigned in the **Pressure Inlet** dialog box will be considered the static temperature at that boundary. If a boundary allows *only a compressible* phase to enter the domain, then the temperature assigned in the **Pressure Inlet** dialog box will be taken as the total temperature (relative/absolute) at that boundary. The total temperature will depend on the **Reference Frame** option selected in the **Pressure Inlet** dialog box.
- For the VOF multiphase model, if a boundary allows a *compressible phase* to enter the domain, then the temperature assigned in the **Pressure Inlet** dialog box will be considered the total temperature at that boundary. The total temperature (relative/absolute) will depend on the **Reference Frame** option chosen in the dialog box. Otherwise, the temperature assigned to the boundary will be considered the static temperature at the boundary.
- For the Eulerian multiphase model, if a boundary allows a mixture of compressible and incompressible phases in the domain, then the temperature of each of the phases will be the total or static temperature, depending on whether the phase is compressible or incompressible.
- Total temperature (relative/absolute) will depend on the **Reference Frame** option chosen in the **Pressure Inlet** dialog box.

6.3.3.1.1.3. Defining the Flow Direction

The flow direction is defined as a unit vector ($\vec{d}$) which is aligned with the local velocity vector, $\vec{v}$. This can be expressed simply as

$$\vec{d}=\frac{\vec{v}}{|\vec{v}|} \tag{6.80}$$

Important

For the inputs in ANSYS Fluent, the flow direction $\vec{d}$ need not be a unit vector, as it will be automatically normalized before it is applied.

Important

For a moving reference frame, the relative flow direction $\vec{d}_r$ is defined in terms of the relative velocity, $\vec{v}_r$. Thus,

$$\vec{d}_r=\frac{\vec{v}_r}{|\vec{v}_r|} \tag{6.81}$$

You can define the flow direction at a pressure inlet explicitly, or you can define the flow to be normal to the boundary. If you choose to specify the direction vector, you can set either the (Cartesian) x, y, and z components, or the (cylindrical) radial, tangential, and axial components.

For moving zone problems calculated using the pressure-based solver, the flow direction will be absolute or relative to the mesh velocity, depending on whether or not the **Absolute** velocity formulation is selected in the **General** task page. For the density-based solver, the flow direction will always be in the absolute frame.

The procedure for defining the flow direction is as follows (refer to Figure 6.20: The Pressure Inlet Dialog Box (p. 309)):

1. Specify the flow direction by selecting **Direction Vector** or **Normal to Boundary** in the **Direction Specification Method** drop-down list.

2. If you selected **Normal to Boundary** in step 1 and you are modeling axisymmetric swirl, enter the appropriate value for the **Tangential-Component of Flow Direction**. If you chose **Normal to Boundary** and your geometry is 3D or 2D without axisymmetric swirl, there are no additional inputs for flow direction.

3. If you selected **Direction Vector** in step 1, and your geometry is 3D, choose **Cartesian (X, Y, Z)**, **Cylindrical(Radial, Tangential, Axial)**, **Local Cylindrical (Radial, Tangential, Axial)**, or **Local Cylindrical Swirl** from the **Coordinate System** drop-down list. Some notes on these selections are provided below:

   - The **Cartesian** coordinate option is based on the Cartesian coordinate system used by the geometry. Enter appropriate values for the **X**, **Y**, and **Z-Component of Flow Direction**.

   - The **Cylindrical** coordinate system uses the axial, radial, and tangential components based on the following coordinate systems:

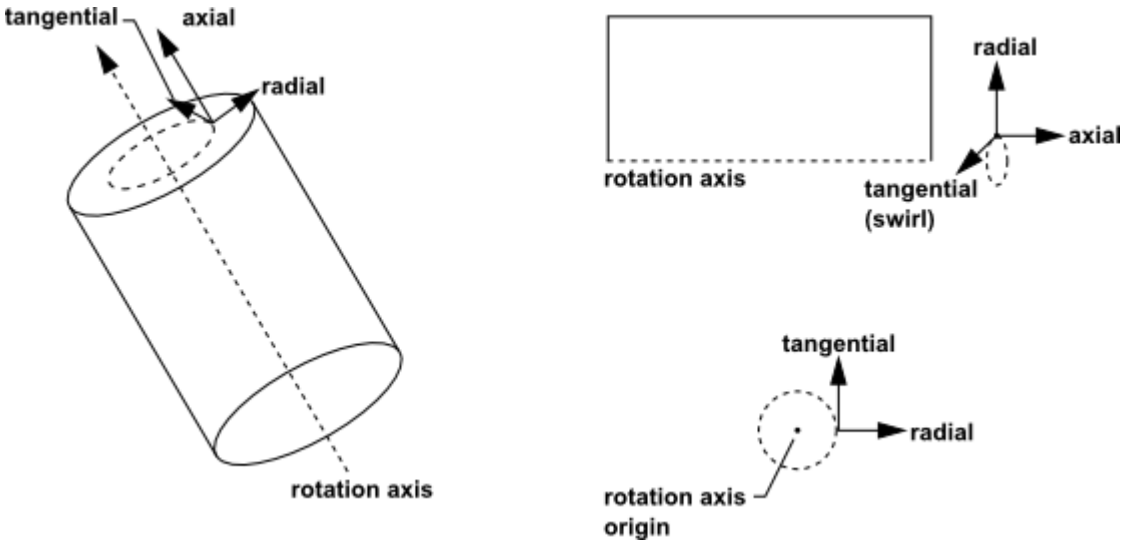
     – For problems involving a single cell zone, the coordinate system is defined by the rotation axis and origin specified in the Fluid Dialog Box (p. 2369).

     – For problems involving multiple zones (for example, multiple reference frames or sliding meshes), the coordinate system is defined by the rotation axis specified in the **Fluid** (or **Solid**) dialog box for the fluid (or solid) zone that is adjacent to the inlet.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For all of the above definitions of the cylindrical coordinate system, positive radial velocities point radially outward from the rotation axis, positive axial velocities are in the direction of the rotation axis vector, and positive tangential velocities are based on the right-hand rule using the positive rotation axis (see Figure 6.21: Cylindrical Velocity Components in 3D, 2D, and Axisymmetric Domains (p. 313)).

Figure 6.21: Cylindrical Velocity Components in 3D, 2D, and Axisymmetric Domains



- The **Local Cylindrical** coordinate system allows you to define a coordinate system specifically for the inlet. When you use the local cylindrical option, you will define the coordinate system right here in the **Pressure Inlet** dialog box. The local cylindrical coordinate system is useful if you have several inlets with different rotation axes. Enter appropriate values for the **Axial**, **Radial**, and **Tangential-Component of Flow Direction**, and then specify the **X**, **Y**, and **Z** components of the Axis Origin and Axis Direction.
- The **Local Cylindrical Swirl** coordinate system option allows you to define a coordinate system specifically for the inlet where the total pressure, swirl velocity, and the components of the velocity in the axial and radial planes are specified. Enter appropriate values for the **Axial** and **Radial-Component of Flow Direction**, and the **Tangential-Velocity**. Specify the **X**, **Y**, and **Z** components of the Axis Origin and Axis Direction. It is recommended that you start your simulation with a smaller swirl velocity and then progressively increase the velocity to obtain a stable solution.

Important

Local Cylindrical Swirl should not be used for open channel boundary conditions and on the mixing plane boundaries while using the mixing plane model.

---

4. If you selected **Direction Vector** in step 1, and your geometry is 2D, define the vector components as follows:
   - For a 2D planar geometry, enter appropriate values for the **X**, **Y**, and **Z-Component of Flow Direction**.
   - For a 2D axisymmetric geometry, enter appropriate values for the **Axial**, **Radial-Component of Flow Direction**.
   - For a 2D axisymmetric swirl geometry, enter appropriate values for the **Axial**, **Radial**, and **Tangential-Component of Flow Direction**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.21: Cylindrical Velocity Components in 3D, 2D, and Axisymmetric Domains (p. 313) shows the vector components for these different coordinate systems.

6.3.3.1.1.4. Defining Static Pressure

The static pressure (termed the **Supersonic/Initial Gauge Pressure**) must be specified if the inlet flow is supersonic *or* if you plan to initialize the solution based on the pressure inlet boundary conditions. Solution initialization is discussed in Initializing the Solution (p. 1591).

Remember that the static pressure value you enter is relative to the operating pressure set in the Operating Conditions Dialog Box (p. 2382). Note the comments in Pressure Inputs and Hydrostatic Head (p. 309) regarding hydrostatic pressure.

The **Supersonic/Initial Gauge Pressure** is ignored by ANSYS Fluent whenever the flow is subsonic, in which case it is calculated from the specified stagnation quantities. If you choose to initialize the solution based on the pressure-inlet conditions, the **Supersonic/Initial Gauge Pressure** will be used in conjunction with the specified stagnation pressure to compute initial values according to the isentropic relations (for compressible flow) or Bernoulli's equation (for incompressible flow). Therefore, for a sub-sonic inlet it should generally be set based on a reasonable estimate of the inlet Mach number (for compressible flow) or inlet velocity (for incompressible flow).

6.3.3.1.1.5. Defining Turbulence Parameters

For turbulent calculations, there are several ways in which you can define the turbulence parameters. Instructions for deciding which method to use and determining appropriate values for these inputs are provided in Determining Turbulence Parameters (p. 303). Turbulence modeling in general is described in Modeling Turbulence (p. 749).

6.3.3.1.1.6. Defining Radiation Parameters

If you are using the P-1, DTRM, DO, surface-to-surface, or MC models, you will set the **Internal Emissivity** and (optionally) **External Black Body Temperature**. See Defining Boundary Conditions for Radiation (p. 869) for details. (The Rosseland radiation model does not require any boundary condition inputs.)

6.3.3.1.1.7. Defining Species Mass or Mole Fractions

If you are modeling species transport, you will set the species mass or mole fractions under **Species Mole Fractions** or **Species Mass Fractions**. For details, see Defining Cell Zone and Boundary Conditions for Species (p. 990).

6.3.3.1.1.8. Defining Non-Premixed Combustion Parameters

If you are using the non-premixed or partially premixed combustion model, you will set the **Mean Mixture Fraction** and **Mixture Fraction Variance** (and the **Secondary Mean Mixture Fraction** and **Secondary Mixture Fraction Variance**, if you are using two mixture fractions), as described in Defining Non-Premixed Boundary Conditions (p. 1079).

6.3.3.1.1.9. Defining Premixed Combustion Boundary Conditions

If you are using the premixed or partially premixed combustion model, you will set the **Progress Variable**, as described in Setting Boundary Conditions for the Progress Variable (p. 1094).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.3.1.1.10. Defining Discrete Phase Boundary Conditions

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the pressure inlet. See Setting Boundary Conditions for the Discrete Phase (p. 1301) for details.

6.3.3.1.1.11. Defining Multiphase Boundary Conditions

If you are using the VOF, mixture, or Eulerian model for multiphase flow, you will need to specify volume fractions for secondary phases and (for some models) additional parameters. See Defining Multiphase Cell Zone and Boundary Conditions (p. 1391) for details.

6.3.3.1.1.12. Defining Open Channel Boundary Conditions

If you are using the VOF model for multiphase flow and modeling open channel flows, you will need to specify the **Free Surface Level**, **Bottom Level**, and additional parameters. See Modeling Open Channel Flows (p. 1410) for details.

6.3.3.2. Default Settings at Pressure Inlet Boundaries

Default settings (in SI) for pressure inlet boundary conditions are as follows:

Gauge Total Pressure	0
Supersonic/Initial Gauge Pressure	0
Total Temperature	300
Direction Specification Method	Normal to Boundary
Turbulent Intensity	5%
Turbulent Viscosity Ratio	10

6.3.3.3. Calculation Procedure at Pressure Inlet Boundaries

The treatment of pressure inlet boundary conditions by ANSYS Fluent can be described as a loss-free transition from stagnation conditions to the inlet conditions. For incompressible flows, this is accomplished by application of the Bernoulli equation at the inlet boundary. In compressible flows, the equivalent isentropic flow relations for an ideal gas are used.

6.3.3.3.1. Incompressible Flow Calculations at Pressure Inlet Boundaries

When flow enters through a pressure inlet boundary, ANSYS Fluent uses the boundary condition pressure you input as the total pressure of the fluid at the inlet plane, p_0. In incompressible flow, the inlet total pressure and the static pressure, p_s, are related to the inlet velocity via Bernoulli's equation:

$$p_0 = p_s + \frac{1}{2}\rho v^2 \tag{6.82}$$

With the resulting velocity magnitude and the flow direction vector you assigned at the inlet, the velocity components can be computed. The inlet mass flow rate and fluxes of momentum, energy, and species can then be computed as outlined in Calculation Procedure at Velocity Inlet Boundaries (p. 322).

For incompressible flows, density at the inlet plane is either constant or calculated as a function of temperature and/or species mass/mole fractions, where the mass or mole fractions are the values you entered as an inlet condition.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If flow exits through a pressure inlet, the total pressure specified is used as the static pressure. For incompressible flows, total temperature is equal to static temperature.

6.3.3.3.2. Compressible Flow Calculations at Pressure Inlet Boundaries

In compressible flows, isentropic relations for an ideal gas are applied to relate total pressure, static pressure, and velocity at a pressure inlet boundary. Your input of total pressure, p_0', at the inlet and the static pressure, p_s', in the adjacent fluid cell are therefore related as

$$\frac{p_0' + p_{op}}{p_s' + p_{op}} = \left(1 + \frac{\gamma - 1}{2} M^2\right)^{\gamma/(\gamma-1)} \tag{6.83}$$

where

$$M \equiv \frac{v}{c} = \frac{v}{\sqrt{\gamma R T_s}} \tag{6.84}$$

c = the speed of sound, and $\gamma = c_p / c_v$. Note that the operating pressure, p_{op}, appears in Equation 6.83 (p. 316) because your boundary condition inputs are in terms of pressure relative to the operating pressure. Given p_0' and p_s', Equation 6.83 (p. 316) and Equation 6.84 (p. 316) are used to compute the velocity magnitude of the fluid at the inlet plane. Individual velocity components at the inlet are then derived using the direction vector components.

For compressible flow, the density at the inlet plane is defined by the ideal gas law in the form

$$\rho = \frac{p_s' + p_{op}}{R T_s} \tag{6.85}$$

For multi-species gas mixtures, the specific gas constant, R, is computed from the species mass or mole fractions, Yi that you defined as boundary conditions at the pressure inlet boundary. The static temperature at the inlet, T_s, is computed from your input of total temperature, T_0, as

$$\frac{T_0}{T_s} = 1 + \frac{\gamma - 1}{2} M^2 \tag{6.86}$$

6.3.4. Velocity Inlet Boundary Conditions

Velocity inlet boundary conditions are used to define the flow velocity, along with all relevant scalar properties of the flow, at flow inlets. In this case, the total (or stagnation) pressure is not fixed but will rise (in response to the computed static pressure) to whatever value is necessary to provide the prescribed velocity distribution. This boundary condition is applicable to incompressible and compressible flows.

Important

Use caution when specifying the velocity inlet boundary condition for a compressible fluid in internal flow modeling. If the flow path is completely choked with no secondary flow path for relief, the velocity inlet boundary condition is not recommended, as it can cause numerical instabilities. In this case, you should switch to the pressure inlet boundary condition or refrain from reaching operating conditions that choke the internal flow. The choking condition can happen in high speed supersonic and transonic flow.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In special instances, a velocity inlet may be used in ANSYS Fluent to define the flow velocity at flow exits. (The scalar inputs are not used in such cases.) In such cases you must ensure that overall continuity is maintained in the domain.

For an overview of flow boundaries, see Flow Inlet and Exit Boundary Conditions (p. 302).

6.3.4.1. Inputs at Velocity Inlet Boundaries

6.3.4.1.1. Summary

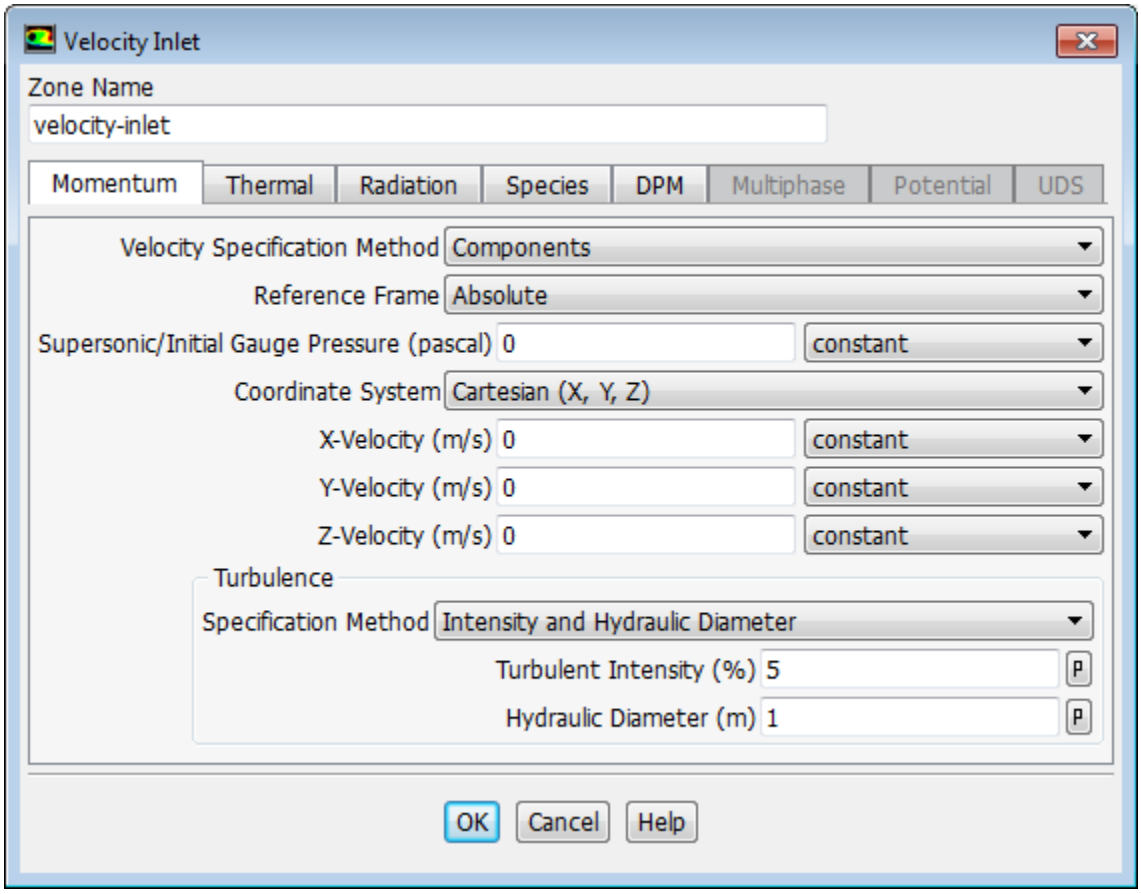
You will enter the following information for a velocity inlet boundary:

- type of reference frame
- velocity magnitude and direction or velocity components
- swirl velocity (for 2D axisymmetric problems with swirl)
- static pressure
- temperature (for energy calculations)
- outflow gauge pressure (for calculations with the density-based solver)
- turbulence parameters (for turbulent calculations)
- radiation parameters (for calculations using the P-1, DTRM, DO, surface-to-surface, or MC models)
- chemical species mass or mole fractions (for species calculations)
- mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
- progress variable (for premixed or partially premixed combustion calculations)
- discrete phase boundary conditions (for discrete phase calculations)
- multiphase boundary conditions (for general multiphase calculations)
- acoustic wave model settings (see Boundary Acoustic Wave Models (p. 405))

All values are entered in the Velocity Inlet Dialog Box (p. 2447) (Figure 6.22: The Velocity Inlet Dialog Box (p. 318)), which is opened from the **Boundary Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 241)). Note that acoustic wave model settings are described in Boundary Acoustic Wave Models (p. 405).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.22: The Velocity Inlet Dialog Box



6.3.4.1.2. Defining the Velocity

You can define the inflow velocity by specifying the velocity magnitude and direction, the velocity components, or the velocity magnitude normal to the boundary. If the cell zone adjacent to the velocity inlet is moving (that is, if you are using a moving reference frame, multiple reference frames, or sliding meshes), you can specify either relative or absolute velocities. For axisymmetric problems with swirl in ANSYS Fluent, you will also specify the swirl velocity.

The procedure for defining the inflow velocity is as follows:

1. Specify the flow direction by selecting **Magnitude and Direction**, **Components**, or **Magnitude, Normal to Boundary** in the **Velocity Specification Method** drop-down list.

2. If the cell zone adjacent to the velocity inlet is moving, you can choose to specify relative or absolute velocities by selecting **Relative to Adjacent Cell Zone** or **Absolute** in the **Reference Frame** drop-down list. If the adjacent cell zone is not moving, **Absolute** and **Relative to Adjacent Cell Zone** will be equivalent, so you need not visit the list.

3. If you are going to set the velocity magnitude and direction or the velocity components, and your geometry is 3D, choose **Cartesian (X, Y, Z)**, **Cylindrical (Radial, Tangential, Axial)**, or **Local Cylindrical (Radial, Tangential, Axial)** from the **Coordinate System** drop-down list. See Defining the Flow Direction (p. 311) for information about Cartesian, cylindrical, and local cylindrical coordinate systems.

4. Set the appropriate velocity parameters, as described below for each specification method.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.4.1.3. Setting the Velocity Magnitude and Direction

If you selected **Magnitude and Direction** as the **Velocity Specification Method** in step 1 above, you will enter the magnitude of the velocity vector at the inflow boundary (the **Velocity Magnitude**) and the direction of the vector:

- If your geometry is 2D non-axisymmetric, or you chose in step 3 to use the **Cartesian** coordinate system, you will define the **X**, **Y**, and (in 3D) **Z-Component of Flow Direction**.
- If your geometry is 2D axisymmetric, or you chose in step 3 to use a **Cylindrical** coordinate system, enter the appropriate values of **Radial**, **Axial**, and (if you are modeling axisymmetric swirl or using cylindrical coordinates) **Tangential-Component of Flow Direction**.
- If you chose in step 3 to use a **Local Cylindrical** coordinate system, enter appropriate values for the **Axial**, **Radial**, and **Tangential-Component of Flow Direction**, and then specify the **X**, **Y**, and **Z** components of the **Axis Origin** and the **Axis Direction**.

Figure 6.21: Cylindrical Velocity Components in 3D, 2D, and Axisymmetric Domains (p. 313) shows the vector components for these different coordinate systems.

6.3.4.1.4. Setting the Velocity Magnitude Normal to the Boundary

If you selected **Magnitude, Normal to Boundary** as the **Velocity Specification Method** in step 1 above, you will enter the magnitude of the velocity vector at the inflow boundary (the **Velocity Magnitude**).

6.3.4.1.5. Setting the Velocity Components

If you selected **Components** as the **Velocity Specification Method** in step 1 above, you will enter the components of the velocity vector at the inflow boundary as follows:

- If your geometry is 2D non-axisymmetric, or you chose in step 3 to use the Cartesian coordinate system, you will define the **X**, **Y**, and (in 3D) **Z-Velocity**.
- If your geometry is 2D axisymmetric without swirl, you will set the **Radial** and **Axial-Velocity**.
- If your model is 2D axisymmetric with swirl, you will set the **Axial**, **Radial**, and **Swirl-Velocity**, and (optionally) the **Angular Velocity**, as described below.
- If you chose in step 3 to use a **Cylindrical** coordinate system, you will set the **Radial**, **Tangential**, and **Axial-Velocity**, and (optionally) the **Angular Velocity**, as described below.
- If you chose in step 3 to use a **Local Cylindrical** coordinate system, you will set the **Radial**, **Tangential**, and **Axial-Velocity**, and (optionally) the **Angular Velocity**, as described below, and then specify the **X**, **Y**, and **Z** component of the **Axis Origin** and the **Axis Direction**.

Important

Remember that positive values for x, y, and z velocities indicate flow in the positive x, y, and z directions. If flow enters the domain in the negative x direction, for example, you will need to specify a negative value for the x velocity. The same holds true for the radial, tangential, and axial velocities. Positive radial velocities point radially out from the axis, positive

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

axial velocities are in the direction of the axis vector, and positive tangential velocities are based on the right-hand rule using the positive axis.

6.3.4.1.6. Setting the Angular Velocity

If you chose **Components** as the **Velocity Specification Method** in step 1 above, and you are modeling axisymmetric swirl, you can specify the inlet **Angular Velocity** Ω in addition to the **Swirl-Velocity**. Similarly, if you chose **Components** as the **Velocity Specification Method** and you chose in step 3 to use a **Cylindrical** or **Local Cylindrical** coordinate system, you can specify the inlet **Angular Velocity** Ω in addition to the **Tangential-Velocity**.

If you specify Ω, v_θ is computed for each face as Ωr, where r is the radial coordinate in the coordinate system defined by the rotation axis and origin. If you specify both the **Swirl-Velocity** and the **Angular Velocity**, or the **Tangential-Velocity** and the **Angular Velocity**, ANSYS Fluent will add v_θ and Ωr to get the swirl or tangential velocity at each face.

6.3.4.1.7. Defining Static Pressure

The static pressure (termed the **Supersonic/Initial Gauge Pressure**) must be specified if the inlet flow is supersonic *or* if you plan to initialize the solution based on the velocity inlet boundary conditions. Solution initialization is discussed in Initializing the Solution (p. 1591).

The **Supersonic/Initial Gauge Pressure** is ignored by ANSYS Fluent whenever the flow is subsonic. If you choose to initialize the flow based on the velocity inlet conditions, the **Supersonic/Initial Gauge Pressure** will be used in conjunction with the specified stagnation quantities to compute initial values according to isentropic relations.

Remember that the static pressure value you enter is relative to the operating pressure set in the Operating Conditions Dialog Box (p. 2382). Note the comments in Pressure Inputs and Hydrostatic Head (p. 309)

6.3.4.1.8. Defining the Temperature

For calculations in which the energy equation is being solved, you will set the static temperature of the flow at the velocity inlet boundary in the **Thermal** tab in the **Temperature** field.

6.3.4.1.9. Defining Outflow Gauge Pressure

If you are using the density-based solver, you can specify an **Outflow Gauge Pressure** for a velocity inlet boundary. If the flow exits the domain at any face on the boundary, that face will be treated as a pressure outlet with the pressure prescribed in the **Outflow Gauge Pressure** field.

6.3.4.1.10. Defining Turbulence Parameters

For turbulent calculations, there are several ways in which you can define the turbulence parameters. Instructions for deciding which method to use and determining appropriate values for these inputs are provided in Determining Turbulence Parameters (p. 303). Turbulence modeling in general is described in Modeling Turbulence (p. 749).

6.3.4.1.11. Defining Radiation Parameters

If you are using the P-1, DTRM, DO, surface-to-surface, or MC models, you will set the **Internal Emissivity** and (optionally) **External Black Body Temperature**. See Defining Boundary Conditions for

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Radiation (p. 869) for details. (The Rosseland radiation model does not require any boundary condition inputs.)

6.3.4.1.12. Defining Species Mass or Mole Fractions

If you are modeling species transport, you will set the species mass or mole fractions under **Species Mole Fractions** or **Species Mass Fractions**. For details, see Defining Cell Zone and Boundary Conditions for Species (p. 990).

6.3.4.1.13. Defining Non-Premixed Combustion Parameters

If you are using the non-premixed or partially premixed combustion model, you will set the **Mean Mixture Fraction** and **Mixture Fraction Variance** (and the **Secondary Mean Mixture Fraction** and **Secondary Mixture Fraction Variance**, if you are using two mixture fractions), as described in Defining Non-Premixed Boundary Conditions (p. 1079).

6.3.4.1.14. Defining Premixed Combustion Boundary Conditions

If you are using the premixed or partially premixed combustion model, you will set the **Progress Variable**, as described in Setting Boundary Conditions for the Progress Variable (p. 1094).

6.3.4.1.15. Defining Discrete Phase Boundary Conditions

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the velocity inlet. See Setting Boundary Conditions for the Discrete Phase (p. 1301) for details.

6.3.4.1.16. Defining Multiphase Boundary Conditions

If you are using the VOF, mixture, or Eulerian model for multiphase flow, you will need to specify volume fractions for secondary phases and (for some models) additional parameters. See Defining Multiphase Cell Zone and Boundary Conditions (p. 1391) for details.

6.3.4.2. Default Settings at Velocity Inlet Boundaries

Default settings (in SI) for velocity inlet boundary conditions are as follows:

Temperature	300
Velocity Magnitude	0
X-Component of Flow Direction	1
Y-Component of Flow Direction	0
Z-Component of Flow Direction	0
X-Velocity	0
Y-Velocity	0
Z-Velocity	0
Turbulent Intensity	5%
Turbulent Viscosity Ratio	10
Outflow Gauge Pressure	0

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.4.3. Calculation Procedure at Velocity Inlet Boundaries

ANSYS Fluent uses your boundary condition inputs at velocity inlets to compute the mass flow into the domain through the inlet and to compute the fluxes of momentum, energy, and species through the inlet. This section describes these calculations for the case of flow entering the domain through the velocity inlet boundary and for the less common case of flow exiting the domain through the velocity inlet boundary.

6.3.4.3.1. Treatment of Velocity Inlet Conditions at Flow Inlets

When your velocity inlet boundary condition defines flow entering the physical domain of the model, ANSYS Fluent uses both the velocity components and the scalar quantities that you defined as boundary conditions to compute the inlet mass flow rate, momentum fluxes, and fluxes of energy and chemical species.

The mass flow rate entering a fluid cell adjacent to a velocity inlet boundary is computed as

$$\dot{m}=\int \rho\vec{v}\cdot d\vec{A} \tag{6.87}$$

Note that only the velocity component normal to the control volume face contributes to the inlet mass flow rate.

6.3.4.3.2. Treatment of Velocity Inlet Conditions at Flow Exits

Sometimes a velocity inlet boundary is used where flow exits the physical domain. This approach might be used, for example, when the flow rate through one exit of the domain is known or is to be imposed on the model.

Important

In such cases you must ensure that overall continuity is maintained in the domain.

In the pressure-based solver, when flow exits the domain through a velocity inlet boundary ANSYS Fluent uses the boundary condition value for the velocity component normal to the exit flow area. It does not use any other boundary conditions that you have input. Instead, all flow conditions except the normal velocity component are assumed to be those of the upstream cell.

In the density-based solver, if the flow exits the domain at any face on the boundary, that face will be treated as a pressure outlet with the pressure prescribed in the **Outflow Gauge Pressure** field.

6.3.4.3.3. Density Calculation

Density at the inlet plane is either constant or calculated as a function of temperature, pressure, and/or species mass/mole fractions, where the mass or mole fractions are the values you entered as an inlet condition.

6.3.5. Mass-Flow Inlet Boundary Conditions

Mass flow boundary conditions can be used in ANSYS Fluent to provide a prescribed mass flow rate or mass flux distribution at an inlet. As with a velocity inlet, specifying the mass flux permits the total pressure to vary in response to the interior solution. This is in contrast to the pressure inlet boundary

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

condition (see Pressure Inlet Boundary Conditions (p. 308)), where the total pressure is fixed while the mass flux varies. The mass-flow inlet is applicable to incompressible and compressible flows.

Important

Use caution when specifying the mass-flow inlet boundary condition for a compressible fluid in internal flow modeling. If the flow path is completely choked with no secondary flow path for relief, the mass-flow inlet boundary condition is not recommended, as it can cause numerical instabilities. In this case, you should switch to the pressure inlet boundary condition or refrain from reaching operating conditions that choke the internal flow. The choking condition can happen in high speed supersonic and transonic flow.

A mass-flow inlet is often used when it is more important to match a prescribed mass flow rate than to match the total pressure of the inflow stream. An example is the case of a small cooling jet that is bled into the main flow at a fixed mass flow rate, while the velocity of the main flow is governed primarily by a (different) pressure inlet/outlet boundary condition pair. A mass-flow inlet boundary condition can also be used as an outflow by specifying the flow direction away from the solution domain.

6.3.5.1. Limitations and Special Considerations

- The adjustment of inlet total pressure might result in a slower convergence, so if both the pressure inlet boundary condition and the mass-flow inlet boundary condition are acceptable choices, you should choose the former.
- It is not necessary to use mass-flow inlets in incompressible flows because when density is constant, velocity inlet boundary conditions will fix the mass flow.

For an overview of flow boundaries, see Flow Inlet and Exit Boundary Conditions (p. 302).

6.3.5.2. Inputs at Mass-Flow Inlet Boundaries

6.3.5.2.1. Summary

You will enter the following information for a mass-flow inlet boundary:

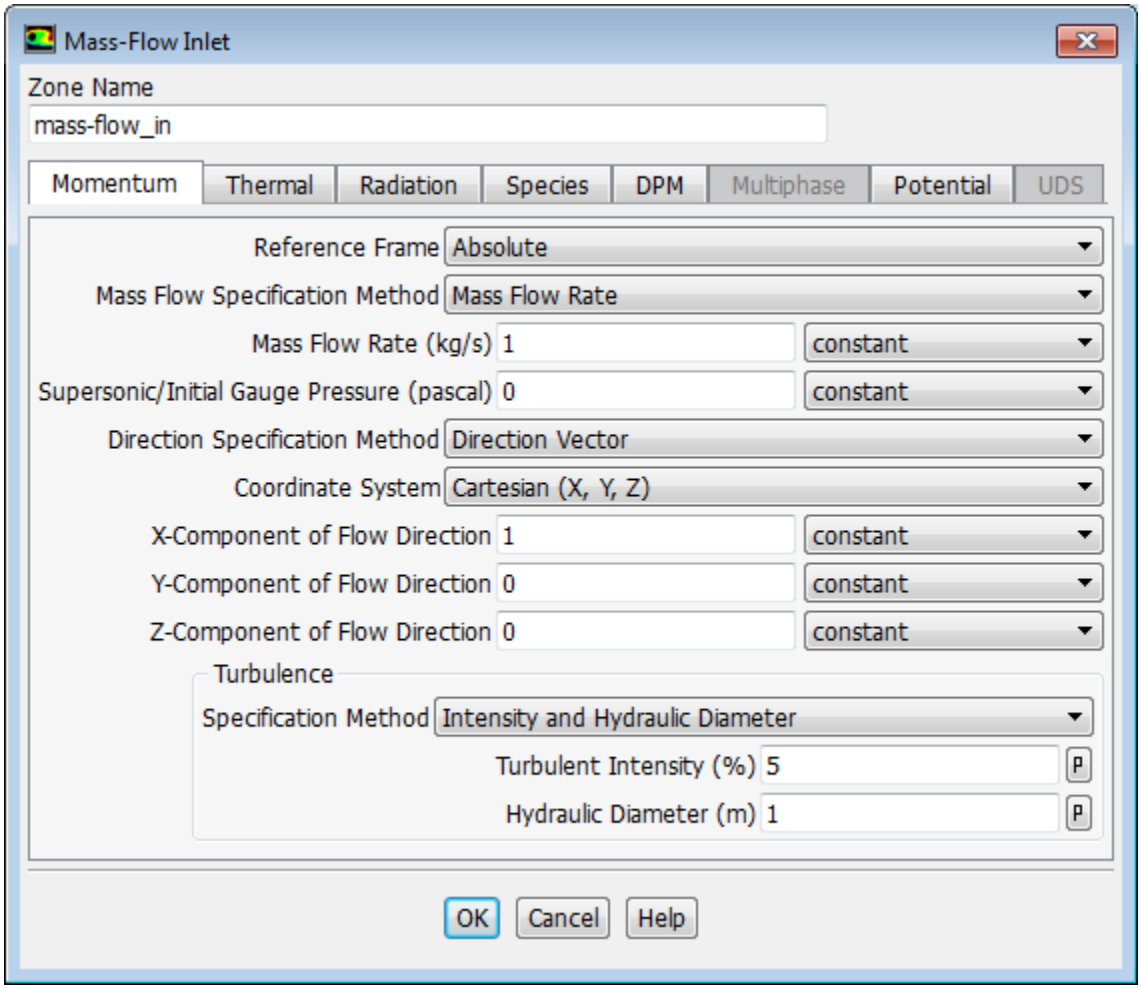
- type of reference frame
- mass flow rate, mass flux, or (primarily for the mixing plane model) mass flux with average mass flux
- total (stagnation) temperature
- static pressure
- flow direction
- turbulence parameters (for turbulent calculations)
- radiation parameters (for calculations using the P-1, DTRM, DO, surface-to-surface, or MC models)
- chemical species mass or mole fractions (for species calculations)
- mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
- progress variable (for premixed or partially premixed combustion calculations)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- discrete phase boundary conditions (for discrete phase calculations)
- open channel flow parameters (for open channel flow calculations using the VOF multiphase model)
- acoustic wave model settings

All values are entered in the Mass-Flow Inlet Dialog Box (p. 2411) (Figure 6.23: The Mass-Flow Inlet Dialog Box (p. 324)), which is opened from the **Boundary Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 241)). Note that open channel boundary condition inputs are described in Modeling Open Channel Flows (p. 1410), and acoustic wave model settings are described in Boundary Acoustic Wave Models (p. 405).

Figure 6.23: The Mass-Flow Inlet Dialog Box



6.3.5.2.2. Selecting the Reference Frame

You will have the option to specify the mass flow boundary conditions either in the absolute or relative reference frame, when the cell zone adjacent to the mass-flow inlet is moving. For such a case, choose **Absolute** (the default) or **Relative to Adjacent Cell Zone** in the **Reference Frame** drop-down list. If the cell zone adjacent to the mass-flow inlet is not moving, both formulations are equivalent.

6.3.5.2.3. Defining the Mass Flow Rate or Mass Flux

You can specify the mass flow rate through the inlet zone and have ANSYS Fluent convert this value to mass flux, or specify the mass flux directly. For cases where the mass flux varies across the boundary,

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

you can also specify an average mass flux; see below for more information about this specification method.

You can define the mass flux or mass flow rate using a profile or a user-defined function.

The inputs for mass flow rate or flux are as follows:

1. Make a selection in the **Mass Flow Specification Method** drop-down list to specify whether you will define a **Mass Flow Rate**, **Mass Flux**, or **Mass Flux with Average Mass Flux**.

2. If you selected **Mass Flow Rate** (the default), enter the prescribed **Mass Flow Rate** when **constant** is selected from the drop-down list. Otherwise, select your hooked user-defined function (UDF) or transient profile.

   Important

   The hooked UDF or transient profile can only be used to provide time-varying specification of mass flow rate. Therefore, the transient solver must be used to run the simulation. Note that the variation of profile with position in space is not applicable with this hookup.

   See `DEFINE_PROFILE` in the Fluent Customization Manual for an example of a mass-flow inlet UDF.

   Important

   Note that for axisymmetric problems, this mass flow rate is the flow rate through the entire (2π-radian) domain, not through a 1-radian slice.

3. If you selected **Mass Flux**, enter the prescribed **Mass Flux**, or select your hooked UDF or profile.

4. If you selected **Mass Flux with Average Mass Flux**, enter the prescribed **Mass Flux** and **Average Mass Flux**.

   Important

   Note that for axisymmetric problems, the **Mass Flux** and **Average Mass Flux** is the flux through a 1-radian slice of the domain.

6.3.5.2.4. More About Mass Flux and Average Mass Flux

As noted previously, you can specify an average mass flux with the mass flux. If, for example, you specify a mass flux profile such that the average mass flux integrated over the zone area is 4.7, but you actually want to have a total mass flux of 5, you can keep the profile unchanged, and specify an average mass flux of 5. ANSYS Fluent will maintain the profile shape but adjust the values so that the resulting mass flux across the boundary is 5.

The mass flux with average mass flux specification method is also used by the mixing plane model described in The Mixing Plane Model (p. 615). If the mass-flow inlet boundary is going to represent one of the mixing planes, then you do *not* need to specify the mass flux or flow rate; you can keep the default **Mass Flow Rate** of 1. When you create the mixing plane later on in the problem setup, ANSYS Fluent will automatically select the **Mass Flux with Average Mass Flux** method in the **Mass-Flow Inlet** dialog

box and set the **Average Mass Flux** to the value obtained by integrating the mass flux profile for the upstream zone. This will ensure that mass is conserved between the upstream zone and the downstream (mass-flow inlet) zone.

6.3.5.2.5. Defining the Total Temperature

Enter the value for the total (stagnation) temperature of the inflow stream in the **Total Temperature** field in the **Thermal** tab.

The total temperature is specified either in the absolute reference frame or relative to the adjacent cell zone, depending on your setting for the **Reference Frame**.

For the Eulerian multiphase model, the total temperature, and mass flux components need to be specified for the individual phases. The **Reference Frame** (**Relative to Adjacent Cell Zone** or **Absolute**) for each of the phases is the same as the reference frame selected for the mixture phase.

Important

Note that you can only set the reference frame for the mixture, however, the total temperature can only be set for the individual phases.

Important

- If the flow is incompressible, then the temperature assigned in the **Mass-Flow Inlet** dialog box is considered to be the static temperature.
- For the mixture multiphase model, if a boundary allows a combination of compressible and incompressible phases to enter the domain, then the temperature assigned in the **Mass-Flow Inlet** dialog box is considered to be the static temperature at that boundary. If a boundary allows *only a compressible phase* to enter the domain, then the temperature assigned in the **Mass-Flow Inlet** dialog box is the total temperature (relative/absolute) at that boundary. The total temperature depends on the **Reference Frame** option selected in the **Mass-Flow Inlet** dialog box.
- For the VOF multiphase model, if a boundary allows a *compressible phase* to enter the domain, then the temperature assigned in the **Mass-Flow Inlet** dialog box is considered to be the total temperature at that boundary. The total temperature (relative/absolute) depends on the **Reference Frame** option chosen in the dialog box. Otherwise, the temperature assigned to the boundary is considered to be the static temperature at the boundary.
- For the Eulerian multiphase model, if a boundary allows a mixture of compressible and incompressible phases in the domain, then the temperature of each of the phases is the total or static temperature, depending on whether the phase is compressible or incompressible. Total temperature (relative/absolute) depends on the **Reference Frame** option chosen in the **Mass-Flow Inlet** dialog box.

6.3.5.2.6. Defining Static Pressure

The static pressure (termed the **Supersonic/Initial Gauge Pressure**) must be specified if the inlet flow is supersonic *or* if you plan to initialize the solution based on the pressure inlet boundary conditions. Solution initialization is discussed in Initializing the Solution (p. 1591).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Supersonic/Initial Gauge Pressure** is ignored by ANSYS Fluent whenever the flow is subsonic. If you choose to initialize the flow based on the mass-flow inlet conditions, the **Supersonic/Initial Gauge Pressure** will be used in conjunction with the specified stagnation quantities to compute initial values according to isentropic relations.

Remember that the static pressure value you enter is relative to the operating pressure set in the Operating Conditions Dialog Box (p. 2382). Note the comments in Pressure Inputs and Hydrostatic Head (p. 309) regarding hydrostatic pressure.

6.3.5.2.7. Defining the Flow Direction

You can define the flow direction at a mass-flow inlet explicitly, or you can define the flow to be normal to the boundary.

The procedure for defining the flow direction is as follows, referring to Figure 6.23: The Mass-Flow Inlet Dialog Box (p. 324):

1. Specify the flow direction by selecting **Direction Vector** or **Normal to Boundary** in the **Direction Specification Method** drop-down list.

2. If you selected **Direction Vector** and your geometry is 2D, go to the next step. If your geometry is 3D, choose **Cartesian (X, Y, Z)**, **Cylindrical (Radial, Tangential, Axial)**, **Local Cylindrical (Radial, Tangential, Axial)**, or **Local Cylindrical Swirl** in the **Coordinate System** drop-down list. See Defining the Flow Direction (p. 311) for information about Cartesian, cylindrical, local cylindrical, and local cylindrical swirl coordinate systems.

3. If you selected **Direction Vector**, set the vector components as follows:

   - If your geometry is 2D non-axisymmetric, or you chose to use a 3D **Cartesian** coordinate system, enter appropriate values for the **X-**, **Y-**, and (in 3D) **Z-Component of Flow Direction**.

   - If your geometry is 2D axisymmetric, or you chose to use a 3D **Cylindrical** coordinate system, enter appropriate values for the **Axial-**, **Radial-**, and (if you are modeling swirl or using cylindrical coordinates) **Tangential-Component of Flow Direction**.

   - If you chose to use a 3D **Local Cylindrical** coordinate system, enter appropriate values for the **Axial-**, **Radial-**, and **Tangential-Component of Flow Direction**, and then specify the **X**, **Y**, and **Z** components of **Axis Origin** and the **Axis Direction**.

   - If you chose to use a 3D **Local Cylindrical Swirl** coordinate system, enter appropriate values for the **Axial-** and **Radial-Component of Flow Direction** in the axial and radial planes, and the **Tangential-Velocity**. Specify the **X**, **Y**, and **Z** components of the **Axis Origin** and the **Axis Direction**.

   Important

   Local Cylindrical Swirl should not be used for open channel boundary conditions and on the mixing plane boundaries, while using the mixing plane model.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. If you selected **Normal to Boundary**, there are no additional inputs for flow direction.

> **Important**
>
> Note that if you are modeling axisymmetric swirl, the flow direction will be normal to the boundary; that is, there will be no swirl component at the boundary for axisymmetric swirl.

6.3.5.2.8. Defining Turbulence Parameters

For turbulent calculations, there are several ways in which you can define the turbulence parameters. Instructions for deciding which method to use and determining appropriate values for these inputs are provided in Determining Turbulence Parameters (p. 303). Turbulence modeling is described in Modeling Turbulence (p. 749).

6.3.5.2.9. Defining Radiation Parameters

If you are using the P-1, DTRM, DO, surface-to-surface, or MC models, you will set the **Internal Emissivity** and (optionally) **External Black Body Temperature**. See Defining Boundary Conditions for Radiation (p. 869) for details. (The Rosseland radiation model does not require any boundary condition inputs.)

6.3.5.2.10. Defining Species Mass or Mole Fractions

If you are modeling species transport, you will set the species mass or mole fractions under **Species Mole Fractions** or **Species Mass Fractions**. For details, see Defining Cell Zone and Boundary Conditions for Species (p. 990).

6.3.5.2.11. Defining Non-Premixed Combustion Parameters

If you are using the non-premixed or partially premixed combustion model, you will set the **Mean Mixture Fraction** and **Mixture Fraction Variance** (and the **Secondary Mean Mixture Fraction** and **Secondary Mixture Fraction Variance**, if you are using two mixture fractions), as described in Defining Non-Premixed Boundary Conditions (p. 1079).

6.3.5.2.12. Defining Premixed Combustion Boundary Conditions

If you are using the premixed or partially premixed combustion model, you will set the **Progress Variable**, as described in Setting Boundary Conditions for the Progress Variable (p. 1094).

6.3.5.2.13. Defining Discrete Phase Boundary Conditions

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the mass-flow inlet. See Setting Boundary Conditions for the Discrete Phase (p. 1301) for details.

6.3.5.2.14. Defining Open Channel Boundary Conditions

If you are using the VOF model for multiphase flow and modeling open channel flows, you will need to specify the **Free Surface Level**, **Bottom Level**, and additional parameters. See Modeling Open Channel Flows (p. 1410) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.5.3. Default Settings at Mass-Flow Inlet Boundaries

Default settings (in SI) for mass-flow inlet boundary conditions are as follows:

Mass Flow Rate	1
Total Temperature	300
Supersonic/Initial Gauge Pressure	0
X-Component of Flow Direction	1
Y-Component of Flow Direction	0
Z-Component of Flow Direction	0
Turbulent Intensity	5%
Turbulent Viscosity Ratio	10

6.3.5.4. Calculation Procedure at Mass-Flow Inlet Boundaries

When mass flow boundary conditions are used for an inlet zone, a velocity is computed for each face in that zone, and this velocity is used to compute the fluxes of all relevant solution variables into the domain. With each iteration, the computed velocity is adjusted so that the correct mass flow value is maintained.

To compute this velocity, your inputs for mass flow rate, flow direction, static pressure, and total temperature are used.

There are two ways to specify the mass flow rate. The first is to specify the total mass flow rate, $\dot{m}$, for the inlet. The second is to specify the mass flux, ρv_n (mass flow rate per unit area). If a total mass flow rate is specified, ANSYS Fluent converts it internally to a uniform mass flux by dividing the mass flow rate by the total inlet area:

$$\rho v_n = \frac{\dot{m}}{A} \tag{6.88}$$

If the direct mass flux specification option is used, the mass flux can be varied over the boundary by using profile files or user-defined functions. If the average mass flux is also specified (either explicitly by you or automatically by ANSYS Fluent), it is used to correct the specified mass flux profile, as described earlier in this section.

Once the value of ρv_n at a given face has been determined, the density, ρ, at the face must be determined in order to find the normal velocity, v_n. The manner in which the density is obtained depends upon whether the fluid is modeled as an ideal gas or not. Each of these cases is examined below.

6.3.5.4.1. Flow Calculations at Mass Flow Boundaries for Ideal Gases

If the fluid is an ideal gas, the static temperature and static pressure are required to compute the density:

$$p = \rho R T \tag{6.89}$$

If the inlet is supersonic, the static pressure used is the value that has been set as a boundary condition. If the inlet is subsonic, the static pressure is extrapolated from the cells inside the inlet face.

The static temperature at the inlet is computed from the total enthalpy, which is determined from the total temperature that has been set as a boundary condition. The total enthalpy is given by

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$h_0(T_0) = h(T) + \frac{1}{2}v^2 \tag{6.90}$$

where the velocity magnitude is related to the mass flow rate given by Equation 6.88 (p. 329) and the known user-specified flow direction vector. Using Equation 6.89 (p. 329) to relate density to the (known) static pressure and (unknown) temperature, Equation 6.90 (p. 330) can be solved to obtain the static temperature.

6.3.5.4.2. Flow Calculations at Mass Flow Boundaries for Incompressible Flows

When you are modeling incompressible flows, the static temperature is equal to the total temperature. The density at the inlet is either constant or readily computed as a function of the temperature and (optionally) the species mass or mole fractions. The velocity is then computed using Equation 6.88 (p. 329).

6.3.5.4.3. Flux Calculations at Mass Flow Boundaries

To compute the fluxes of all variables at the inlet, the flux velocity, v_n, is used along with the inlet value of the variable in question. For example, the flux of mass is ρv_n, and the flux of turbulence kinetic energy is $\rho k v_n$. These fluxes are used as boundary conditions for the corresponding conservation equations during the course of the solution.

6.3.6. Mass-Flow Outlet Boundary Conditions

Mass-flow outlet boundaries can be used in ANSYS Fluent to pump flow out of the domain at a prescribed mass flow rate or mass flux distribution. A mass-flow outlet is often used when it is more important to match a prescribed mass flow rate than to match the static pressure of the outflow stream.

6.3.6.1. Limitations

Mass-flow outlet boundaries have the following limitations:

- It is supported with single-phase flow only, and so is not supported with the VOF, mixture, and Eulerian multiphase models in the pressure-based solver.
- It is not supported with the Wet Steam model in the density-based solver.

6.3.6.2. Inputs at Mass-Flow Outlet Boundaries

6.3.6.2.1. Summary

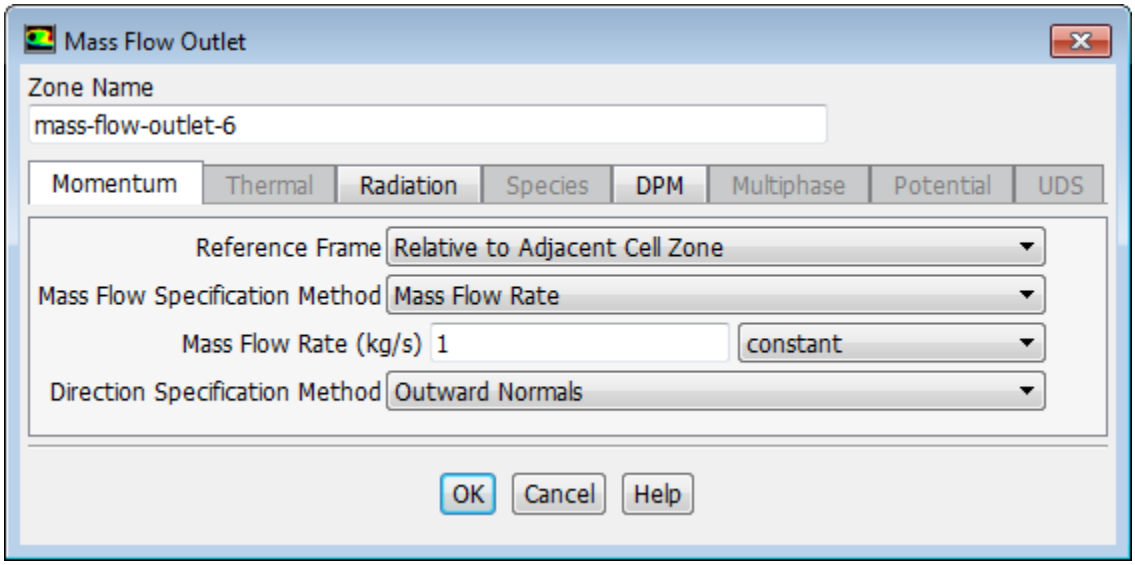
You will enter the following information for a mass-flow outlet boundary:

- mass flow rate, mass flux, or mass flux with average mass flux
- flow direction
- radiation parameters (for calculations using the P-1, DTRM, DO, surface-to-surface, or MC models)
- discrete phase boundary conditions (for discrete phase calculations)

All values are entered in the Mass Flow Outlet Dialog Box (p. 2417) (Figure 6.24: The Mass-Flow Outlet Dialog Box (p. 331)), which is opened from the **Boundary Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 241)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.24: The Mass-Flow Outlet Dialog Box



6.3.6.2.2. Selecting the Reference Frame

Mass-flow outlets only support the **Relative to Adjacent Cell Zone** specification for the **Reference Frame**.

6.3.6.2.3. Defining the Mass Flow Rate or Mass Flux

You can specify the mass flow rate through the outlet zone and have ANSYS Fluent convert this value to mass flux, or specify the mass flux directly. For cases where the mass flux varies across the boundary, you can also specify an average mass flux; see below for more information about this specification method.

You can define the mass flux or mass flow rate using a profile or a user-defined function.

The inputs for mass flow rate or flux are as follows:

1. Make a selection in the **Mass Flow Specification Method** drop-down list to specify whether you will define a **Mass Flow Rate**, **Mass Flux**, or **Mass Flux with Average Mass Flux**.

2. If you selected **Mass Flow Rate** (the default), enter the prescribed **Mass Flow Rate** when **constant** is selected from the drop-down list. Otherwise, select your hooked user-defined function (UDF) or transient profile.

   Important

   The hooked UDF or transient profile can only be used to provide time-varying specification of mass flow rate. Therefore, the transient solver must be used to run the simulation.
   Note that the variation of profile with position in space is not applicable with this hookup.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See DEFINE_PROFILE in the Fluent Customization Manual for an example of a mass-flow outlet UDF.

Important

Note that for axisymmetric problems, this mass flow rate is the flow rate through the entire (2π-radian) domain, not through a 1-radian slice.

3. If you selected **Mass Flux**, enter the prescribed **Mass Flux**, or select your hooked UDF or profile.
4. If you selected **Mass Flux with Average Mass Flux**, enter the prescribed **Mass Flux** and **Average Mass Flux**.

Important

Note that for axisymmetric problems, the **Mass Flux** and **Average Mass Flux** is the flux through a 1-radian slice of the domain.

6.3.6.2.4. More About Mass Flux and Average Mass Flux

As noted previously, you can specify an average mass flux with the mass flux. If, for example, you specify a mass flux profile such that the average mass flux integrated over the zone area is 4.7, but you actually want to have a total mass flux of 5, you can keep the profile unchanged, and specify an average mass flux of 5. ANSYS Fluent will maintain the profile shape but adjust the values so that the resulting mass flux across the boundary is 5.

6.3.6.2.5. Defining the Flow Direction

You can define the flow direction at a mass-flow outlet explicitly, or you can define the flow to be leaving normal to the boundary.

The procedure for defining the flow direction is as follows, referring to Figure 6.24: The Mass-Flow Outlet Dialog Box (p. 331):

1. Specify the flow direction by selecting **Direction Vector** or **Outward Normals** in the **Direction Specification Method** drop-down list.
2. If you selected **Direction Vector** and your geometry is 2D, go to the next step. If your geometry is 3D, choose **Cartesian (X, Y, Z)**, **Cylindrical (Radial, Tangential, Axial)**, **Local Cylindrical (Radial, Tangential, Axial)**, or **Local Cylindrical Swirl** in the **Coordinate System** drop-down list. See Defining the Flow Direction (p. 311) for information about Cartesian, cylindrical, local cylindrical, and local cylindrical swirl coordinate systems.
3. If you selected **Direction Vector**, set the vector components as follows:
   - If your geometry is 2D non-axisymmetric, or you chose to use a 3D **Cartesian** coordinate system, enter appropriate values for the **X-**, **Y-**, and (in 3D) **Z-Component of Flow Direction**.
   - If your geometry is 2D axisymmetric, or you chose to use a 3D **Cylindrical** coordinate system, enter appropriate values for the **Axial-**, **Radial-**, and (if you are modeling swirl or using cylindrical coordinates) **Tangential-Component of Flow Direction**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you chose to use a 3D **Local Cylindrical** coordinate system, enter appropriate values for the **Axial-**, **Radial-**, and **Tangential-Component of Flow Direction**, and then specify the **X**, **Y**, and **Z** components of **Axis Origin** and the **Axis Direction**.
- If you chose to use a 3D **Local Cylindrical Swirl** coordinate system, enter appropriate values for the **Axial-** and **Radial-Component of Flow Direction** in the axial and radial planes, and the **Tangential-Velocity**. Specify the **X**, **Y**, and **Z** components of the **Axis Origin** and the **Axis Direction**.

Important

Note that if you are modeling axisymmetric swirl, the flow direction will be normal to the boundary; that is, there will be no swirl component at the boundary for axisymmetric swirl.

4. If **Outward Normals** is selected, then the mass flow boundary will be pumping flow out of the domain normal to boundary at the prescribed rate or flux. If the mass flow rate is specified, then by default, the fluxes on the boundary will be allowed to vary to preserve the flow profile out of the domain. At convergence, the total mass flow rate should match the specified value.

Important

Note that if constant mass flux is needed rather than the default variable fluxes to preserve the profiles, then you can do so via the text command `define/boundary-conditions/bc-settings/mass-flow`. Answer `no` when asked to `preserve profile while flow leaves`.

Note

The mass-flow outlet boundary condition is designed to be strictly used only when flow is leaving the domain; it should not be used for mixed flow conditions (where flow is flowing in and out) or when flow is changing directions.

6.3.6.2.6. Defining Radiation Parameters

If you are using the P-1, DTRM, DO, surface-to-surface, or MC models, you will set the **Internal Emissivity** and (optionally) **External Black Body Temperature**. See Defining Boundary Conditions for Radiation (p. 869) for details. (The Rosseland radiation model does not require any boundary condition inputs.)

6.3.6.2.7. Defining Discrete Phase Boundary Conditions

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the mass-flow outlet. See Setting Boundary Conditions for the Discrete Phase (p. 1301) for details.

6.3.6.3. Default Settings at Mass-Flow Outlet Boundaries

Default settings (in SI) for mass-flow outlet boundary conditions are as follows:

Mass Flow Rate	1

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Direction Specification Method	Outward Normals

6.3.6.4. Calculation Procedure at Mass-Flow Outlet Boundaries

There are two ways to specify the mass flow rate. The first is to specify the total mass flow rate, $\dot{m}$, for the outlet. The second is to specify the mass flux, ρv_n (mass flow rate per unit area). If a total mass flow rate is specified, ANSYS Fluent converts it internally to a uniform mass flux by dividing the mass flow rate by the total outlet area:

$$\rho v_n = \frac{\dot{m}}{A} \tag{6.91}$$

If the direct mass flux specification option is used, the mass flux can be varied over the boundary by using profile files or user-defined functions. If the average mass flux is also specified, it is used to correct the specified mass flux profile, as described previously in this section.

Once the value of ρv_n at a given face has been determined, the density, ρ, at the face must be determined in order to find the normal velocity, v_n.

Boundary velocities are scaled to achieve the specified value of mass flow rate or mass flux. When the mass-flow outlet is used with the profile preserving feature, a scaling factor of the specified mass flow rate over the computed mass flow rate at the boundary is used to scale the normal face velocities at the boundary. The other velocity components will be extrapolated from the interior. Flow variables such as pressure, temperature, species, or other scalar quantities will be also extrapolated from adjacent cell centers.

6.3.7. Inlet Vent Boundary Conditions

Inlet vent boundary conditions are used to model an inlet vent with a specified loss coefficient, flow direction, and ambient (inlet) pressure and temperature.

6.3.7.1. Inputs at Inlet Vent Boundaries

You will enter the following information for an inlet vent boundary:

- type of reference frame
- total (stagnation) pressure
- total (stagnation) temperature
- flow direction
- static pressure
- turbulence parameters (for turbulent calculations)
- radiation parameters (for calculations using the P-1, DTRM, DO, surface-to-surface, or MC models)
- chemical species mass or mole fractions (for species calculations)
- mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
- progress variable (for premixed or partially premixed combustion calculations)

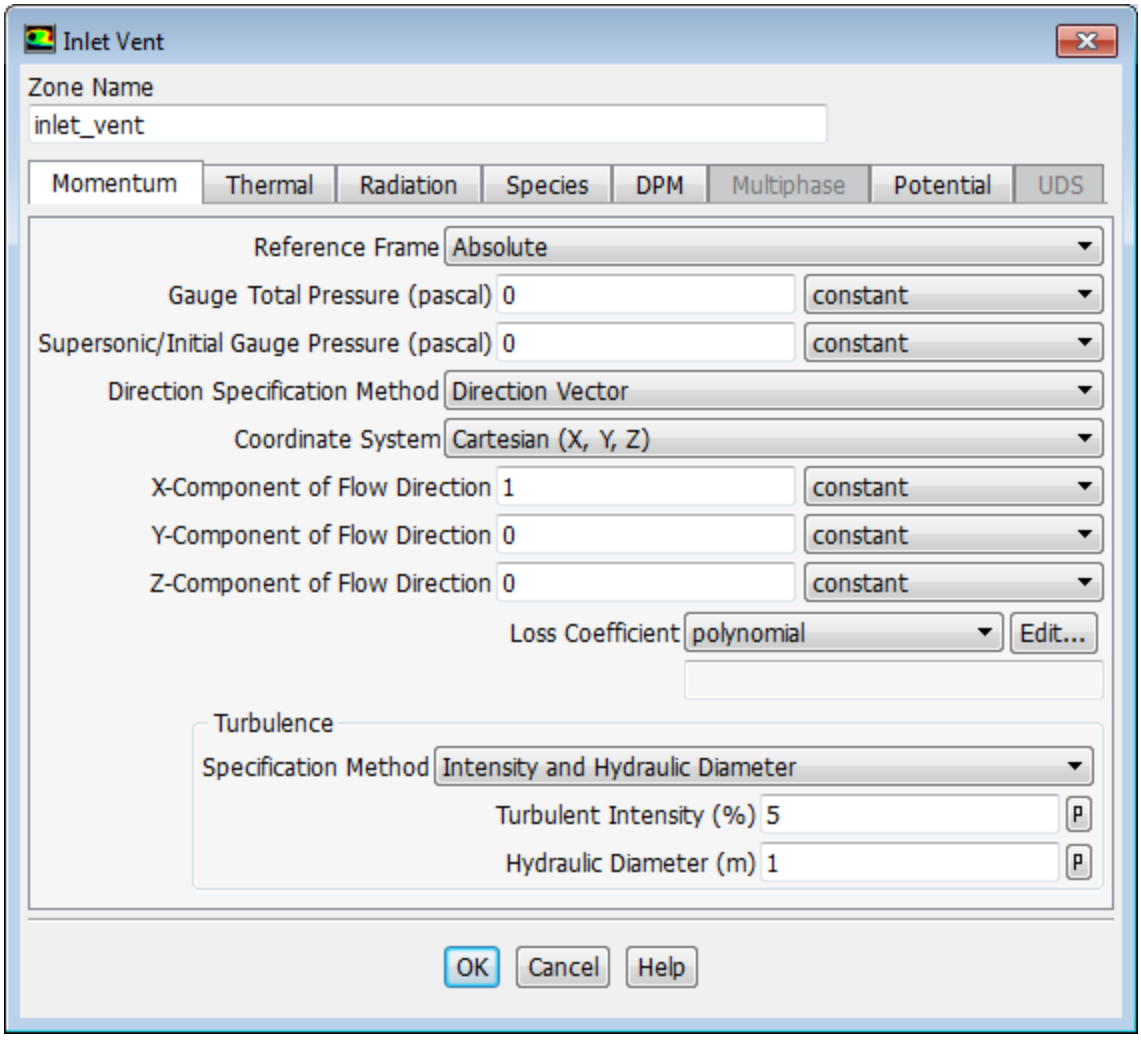
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- discrete phase boundary conditions (for discrete phase calculations)
- multiphase boundary conditions (for general multiphase calculations)
- loss coefficient
- open channel flow parameters (for open channel flow calculations using the VOF multiphase model)

All values are entered in the Inlet Vent Dialog Box (p. 2400) (Figure 6.25: The Inlet Vent Dialog Box (p. 335)), which is opened from the **Boundary Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 241)).

The first 12 items listed above are specified in the same way that they are specified at pressure inlet boundaries. See Inputs at Pressure Inlet Boundaries (p. 308) for details. Specification of the loss coefficient is described here. Open channel boundary condition inputs are described in Modeling Open Channel Flows (p. 1410).

Figure 6.25: The Inlet Vent Dialog Box



6.3.7.1.1. Specifying the Loss Coefficient

An inlet vent is considered to be infinitely thin, and the pressure drop through the vent is assumed to be proportional to the dynamic head of the fluid, with an empirically determined loss coefficient that

you supply. That is, the pressure drop, Δp, varies with the normal component of velocity through the vent, v, as follows:

$$\Delta p = k_L \frac{1}{2} \rho v^2 \tag{6.92}$$

where ρ is the fluid density, and k_L is the non-dimensional loss coefficient.

Important

Δp is the pressure drop in the direction of the flow; therefore the vent will appear as a resistance even in the case of backflow.

You can define the **Loss-Coefficient** across the vent as a **constant**, **polynomial**, **piecewise-linear**, or **piecewise-polynomial** function of the normal velocity. The dialog boxes for defining these functions are the same as those used for defining temperature-dependent properties. See Defining Properties Using Temperature-Dependent Functions (p. 475) for details.

6.3.8. Intake Fan Boundary Conditions

Intake fan boundary conditions are used to model an external intake fan with a specified pressure jump, flow direction, and ambient (intake) pressure and temperature.

Note

The intake fan boundary condition considers only momentum. Diffusion processes are not considered.

6.3.8.1. Inputs at Intake Fan Boundaries

You will enter the following information for an intake fan boundary:

- type of reference frame
- total (stagnation) pressure
- total (stagnation) temperature
- flow direction
- static pressure
- turbulence parameters (for turbulent calculations)
- radiation parameters (for calculations using the P-1, DTRM, DO, surface-to-surface, or MC models)
- chemical species mass or mole fractions (for species calculations)
- mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
- progress variable (for premixed or partially premixed combustion calculations)
- discrete phase boundary conditions (for discrete phase calculations)

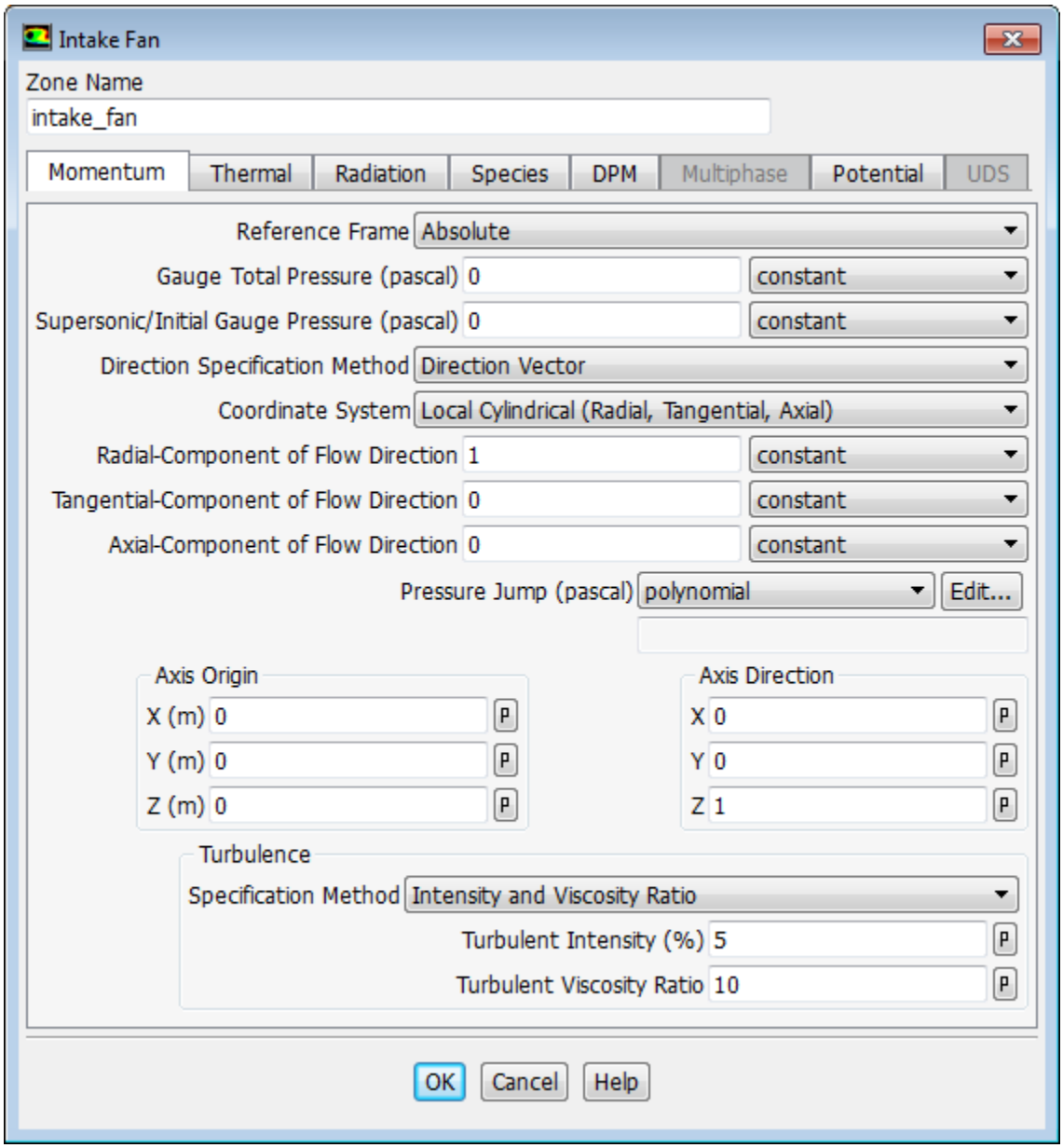
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- multiphase boundary conditions (for general multiphase calculations)
- pressure jump
- open channel flow parameters (for open channel flow calculations using the VOF multiphase model)

All values are entered in the Intake Fan Dialog Box (p. 2406) (shown in Figure 6.26: The Intake Fan Dialog Box (p. 337)), which is opened from the **Boundary Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 241)).

The first 12 items listed above are specified in the same way that they are specified at pressure inlet boundaries. See Inputs at Pressure Inlet Boundaries (p. 308) for details. Specification of the pressure jump is described here. Open channel boundary condition inputs are described in Modeling Open Channel Flows (p. 1410).

Figure 6.26: The Intake Fan Dialog Box



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.8.1.1. Specifying the Pressure Jump

An intake fan is considered to be infinitely thin, and the discontinuous pressure rise across it is specified as a function of the velocity through the fan. In the case of reversed flow, the fan is treated like an outlet vent with a loss coefficient of unity.

You can define the **Pressure-Jump** across the fan as a **constant**, **polynomial**, **piecewise-linear**, or **piecewise-polynomial** function of the normal velocity. The dialog boxes for defining these functions are the same as those used for defining temperature-dependent properties. See Defining Properties Using Temperature-Dependent Functions (p. 475) for details.

6.3.9. Pressure Outlet Boundary Conditions

Pressure outlet boundary conditions require the specification of a static (gauge) pressure at the outlet boundary. The value of the specified static pressure is used only while the flow is subsonic. Should the flow become locally supersonic, the specified pressure will no longer be used; pressure will be extrapolated from the flow in the interior. All other flow quantities are extrapolated from the interior.

A set of "backflow" conditions is also specified should the flow reverse direction at the pressure outlet boundary during the solution process. Convergence difficulties will be minimized if you specify realistic values for the backflow quantities.

Several options in ANSYS Fluent exist, where a radial equilibrium outlet boundary condition can be used (see Defining Static Pressure (p. 339) and a target mass flow rate for pressure outlets (see Target Mass Flow Rate Option (p. 344) for details) can be specified.

For an overview of flow boundaries, see Flow Inlet and Exit Boundary Conditions (p. 302).

6.3.9.1. Inputs at Pressure Outlet Boundaries

6.3.9.1.1. Summary

You will enter or select the following information for a pressure outlet boundary:

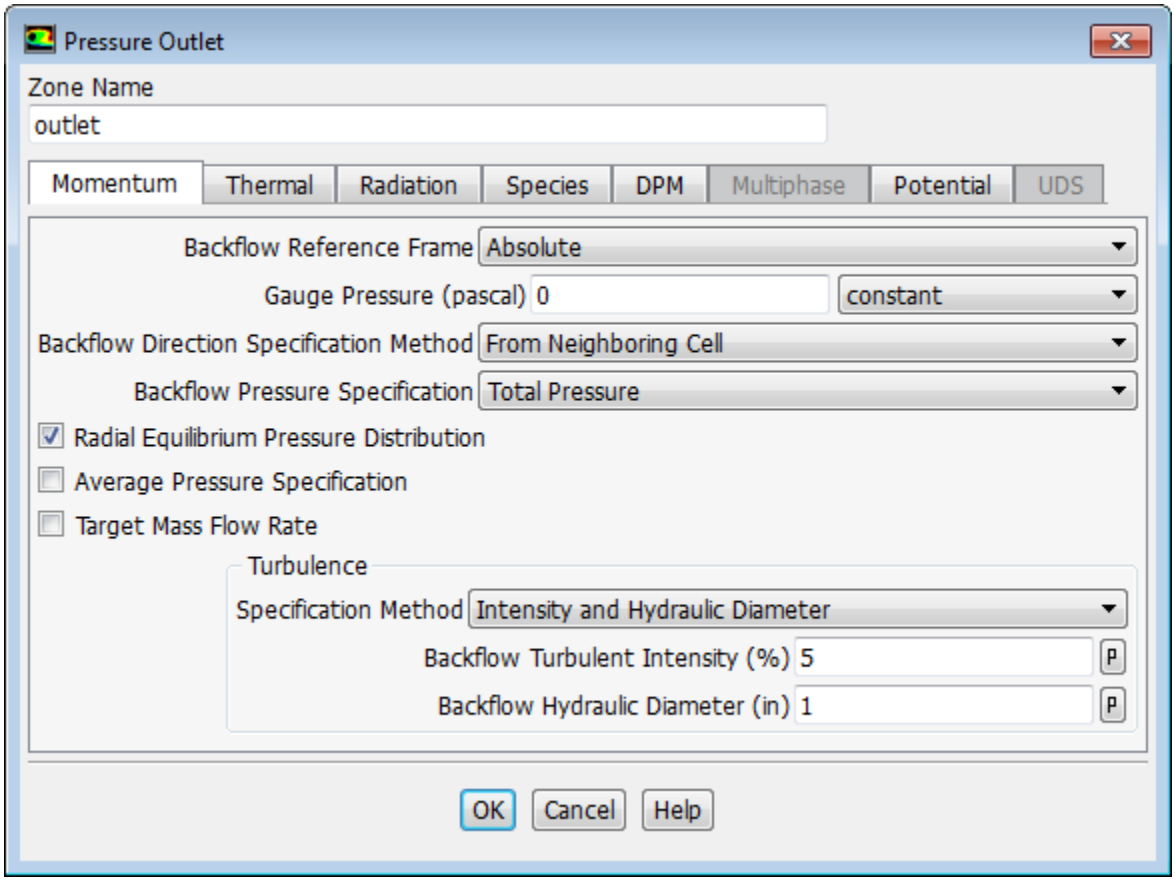
- static pressure
- backflow conditions
  - total (stagnation) temperature (for energy calculations)
  - backflow pressure specification
  - backflow direction specification method
  - turbulence parameters (for turbulent calculations)
  - chemical species mass or mole fractions (for species calculations)
  - mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
  - progress variable (for premixed or partially premixed combustion calculations)
  - multiphase boundary conditions (for general multiphase calculations)
  - type of backflow reference frame (not available for multiphase flows or with the acoustic wave model)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- radiation parameters (for calculations using the P-1, DTRM, DO, surface-to-surface, or MC models)
- discrete phase boundary conditions (for discrete phase calculations)
- open channel flow parameters (for open channel flow calculations using the VOF multiphase model)
- radial equilibrium pressure distribution
- average pressure specification (not available for multiphase flows)
- target mass flow rate (not available for multiphase flows)
- acoustic wave model settings

All values are entered in the Pressure Outlet Dialog Box (p. 2439) (Figure 6.27: The Pressure Outlet Dialog Box (p. 339)), which is opened from the **Boundary Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 241)). Note that open channel boundary condition inputs are described in Modeling Open Channel Flows (p. 1410), and acoustic wave model settings are described in Boundary Acoustic Wave Models (p. 405).

Figure 6.27: The Pressure Outlet Dialog Box



6.3.9.1.2. Defining Static Pressure

To set the static pressure at the pressure outlet boundary, enter the appropriate value for **Gauge Pressure** in the **Pressure Outlet** dialog box. This value will be used when the flow is locally subsonic. Should the flow become locally supersonic, the pressure will be extrapolated from the upstream condi-

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tions. By default the subsonic and supersonic local flow condition at the boundary is determined based on the local normal velocity component.

Remember that the static pressure value you enter is relative to the operating pressure set in the Operating Conditions Dialog Box (p. 2382). Refer to Pressure Inputs and Hydrostatic Head (p. 309) regarding hydrostatic pressure.

ANSYS Fluent also provides an option to use a radial equilibrium outlet boundary condition. This option is used to model the exit flow in turbomachinery flow problems. To turn on this option, enable **Radial Equilibrium Pressure Distribution**. When this feature is active, the specified gauge pressure applies only to the position of minimum radius (relative to the axis of rotation) at the boundary. The static pressure on the rest of the zone is calculated from the assumption that radial velocity is negligible, so that the pressure gradient is given by

$$\frac{\partial p}{\partial r} = \frac{\rho v_\theta^2}{r} \tag{6.93}$$

where r is the distance from the axis of rotation and v_θ is the tangential velocity. Note that this boundary condition can be used even if the rotational velocity is zero. For example, it could be applied to the calculation of the flow through an annulus containing guide vanes.

Important

Note that the radial equilibrium outlet condition is available only for 3D and axisymmetric swirl calculations.

ANSYS Fluent also provides an option to use an **Average Pressure Specification** method at the pressure outlet boundary. This option allows the pressure along the outlet boundary to vary, but maintain an average equivalent to the specified value in the **Gauge Pressure** input field. The pressure variation allowed in this boundary implementation slightly diminishes the reflectivity of the boundary as compared with the default uniform pressure specification. In the density-based solver two averaging methods are available, **Strong Averaging** and **Weak Averaging**. The average pressure specification in the pressure-based solver is the same as the strong pressure averaging in the density-based solver. For more details, see Calculation Procedure at Pressure Outlet Boundaries (p. 342).

6.3.9.1.3. Defining Backflow Conditions

Backflow properties consistent with the models you are using will appear in the **Pressure Outlet** dialog box. The specified values will be used only if flow is pulled in through the outlet.

- The **Backflow Total Temperature** (in the **Thermal** tab) should be set for problems involving energy calculation.
- When the direction of the backflow re-entering the computational domain is known, and deemed to be relevant to the flow field solution, you can specify it choosing one of the options available in the **Backflow Direction Specification Method** drop-down list. The default value for this field is **Normal to Boundary**, and requires no further input. If you choose **Direction Vector**, the dialog box will expand to show the inputs for the components of the direction vector for the backflow, and if you are running the 3D version of ANSYS Fluent, the dialog box will display a **Coordinate System** drop-down list. If you choose **From Neighboring Cell**, ANSYS Fluent will determine the direction of the backflow using the direction of the flow in the cell layer adjacent to the pressure outlet.
- For turbulent calculations, there are several ways in which you can define the turbulence parameters. Instructions for deciding which method to use in determining appropriate values for these inputs are provided in

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Determining Turbulence Parameters (p. 303). Turbulence modeling in general is described in Modeling Turbulence (p. 749).

- If you are modeling species transport, you will set the backflow species mass or mole fractions under **Species Mass Fractions** or **Species Mole Fractions**. For details, see Defining Cell Zone and Boundary Conditions for Species (p. 990).
- If you are modeling combustion using the non-premixed or partially premixed combustion model, you will set the backflow mixture fraction and variance values. See Defining Non-Premixed Boundary Conditions (p. 1079) for details.
- If you are modeling combustion using the premixed or partially premixed combustion model, you will set the backflow **Progress Variable** value. See Setting Boundary Conditions for the Progress Variable (p. 1094) for details.
- If you are using the VOF, mixture, or Eulerian model for multiphase flow, you will need to specify volume fractions for secondary phases and (for some models) additional parameters. See Defining Multiphase Cell Zone and Boundary Conditions (p. 1391) for details.
- If backflow occurs,

  1. The pressure you specified as the **Gauge Pressure** will be used to determine the total pressure or the static pressure (based on your selection from the **Backflow Pressure Specification** drop-down list), so you need not specify a backflow pressure value explicitly. When **Total Pressure** is selected for **Backflow Pressure Specification**, boundary face pressure values are calculated by combining the **Gauge Pressure** with a dynamic contribution that is based on the velocity in the adjacent cell zone; when **Static Pressure** is selected for **Backflow Pressure Specification**, the specified **Gauge Pressure** value is directly imposed as the boundary face pressure.

  2. The flow direction will be based on your selection from the **Backflow Direction Specification Method** drop-down list.

  3. The total temperature, total pressure, and flow directions are in *absolute* or *relative to the adjacent cell zone* reference frames, based on the **Backflow Reference Frame** setting specified in the **Pressure Outlet** dialog box.

  If the cell zone adjacent to the pressure outlet is moving (that is, if you are using a moving reference frame, multiple reference frames, mixing planes, or sliding meshes) and you are using the pressure-based solver, the velocity in the dynamic contribution to total pressure (see Equation 6.78 (p. 310)) will be absolute or relative to the motion of the cell zone, depending on whether or not the **Absolute** velocity formulation is selected in the **General** task page. For the density-based solver, the velocity in Equation 6.78 (p. 310) (or the Mach number in Equation 6.79 (p. 310)) is always in the absolute frame.

Important

Even if no backflow is expected in the converged solution, you should always set realistic values to minimize convergence difficulties in the event that backflow does occur during the calculation.

The **Backflow Reference Frame** option is not available for all multiphase models or non-reflecting boundary condition options.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.9.1.4. Defining Radiation Parameters

If you are using the P-1, DTRM, DO, surface-to-surface, or MC models, you will set the **Internal Emissivity** and (optional) **External Black Body Temperature Method**. See Defining Boundary Conditions for Radiation (p. 869) for details. (The Rosseland radiation model does not require any boundary condition inputs.)

6.3.9.1.5. Defining Discrete Phase Boundary Conditions

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the pressure outlet. See Setting Boundary Conditions for the Discrete Phase (p. 1301) for details.

6.3.9.1.6. Defining Open Channel Boundary Conditions

If you are using the VOF model for multiphase flow and modeling open channel flows, you will need to specify the **Free Surface Level**, **Bottom Level**, and additional parameters. See Modeling Open Channel Flows (p. 1410) for details.

6.3.9.2. Default Settings at Pressure Outlet Boundaries

Default settings (in SI) for pressure outlet boundary conditions are as follows:

Gauge Pressure	0
Backflow Total Temperature	300
Backflow Turbulent Intensity	5%
Backflow Turbulent Viscosity Ratio	10

6.3.9.3. Calculation Procedure at Pressure Outlet Boundaries

At pressure outlets, if the flow exiting the boundary is subsonic, then ANSYS Fluent uses the pressure specified in the **Pressure Outlet** dialog box as the static pressure of the fluid at the outlet plane, P_s, and extrapolates all other conditions from the interior of the domain. However, if the flow exiting the boundary becomes supersonic, then the face pressure values are extrapolated from the interior cell pressure.

6.3.9.3.1. Average Pressure Specification

When this option is selected, the exit pressure is not kept constant over the pressure outlet boundary, but instead is allowed to vary across the boundary while maintaining an average boundary pressure close to the specified static exit pressure. This option can be used for pressure specified as a constant or as a profile (for example, in turbomachinery cases with radial, axial, or radial equilibrium pressure distributions).

There are two implementations of the **Average Pressure Specification**. These are **Strong Averaging** and **Weak Averaging**. The strong averaging approach is available in both the density-based and the pressure-based solver. The weak averaging approach is available only in the density-based solver.

Note

- The **Average Pressure Specification** option is not available with multiphase flows.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When using the density-based solver, **Strong Averaging** is the recommended method and is used by default. The weak averaging method is maintained primarily for legacy cases.
- When one of the **Acoustic Wave Models** is active, the Average Pressure Specification option is not offered because the pressure will be obtained based on the **Acoustic Wave Model** chosen.
- Using the Average Pressure Specification option in unsteady simulations can result in nonphysical pressure feedback and is not recommended.

6.3.9.3.1.1. Strong Averaging

In the **Strong Averaging** approach, the face pressure value P_f for subsonic exit flow is computed using the following expression:

$$P_f = P_e + \left(1 - F\right)\left(P_c - P_{c_avg}\right) \tag{6.94}$$

where

P_c = interior cell pressure at neighboring exit face, f

P_{c_avg} = averaged interior cell pressure at a boundary

P_e = specified exit pressure

F = pressure blending factor; $F=0$ recovers the fully averaged pressure, $F=1$ recovers the specified pressure.

When the strong averaging option is used with a profile specification method (radial, axial, or radial equilibrium pressure distribution) then the outlet boundary will be subdivided into radial or axial containers called "bins", and the above averaging procedure is applied to each bin.

You can use the following TUI command to change the blending factor and the number of bins:

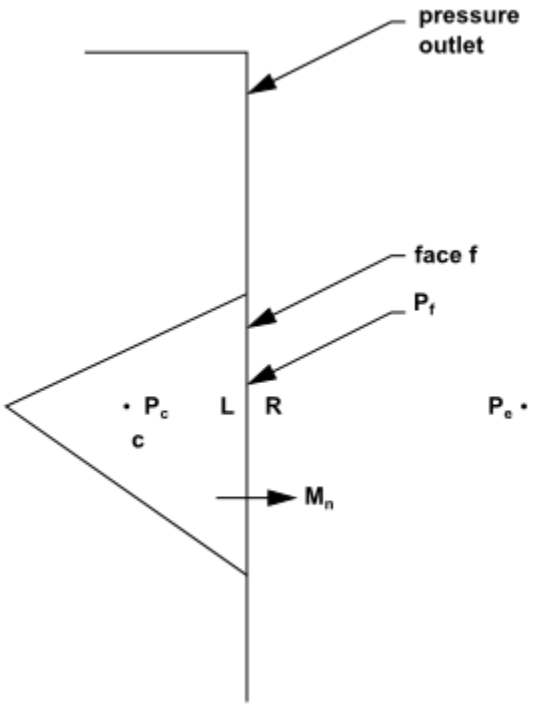
```
/define/boundary-conditions/bc-settings/pressure-outlet
```

6.3.9.3.1.2. Weak Averaging

When the weak averaging method is used and the flow is subsonic, the pressure at the faces of the outlet boundary is computed using a weighted average of the left and right state of the face boundary. This weighting is a blend of fifth-order polynomials based on the exit face normal Mach number [61] (p. 2890). Therefore, the face pressure P_f is a function of (P_c, P_e, M_n), where P_c is the interior cell pressure neighboring the exit face f, P_e is the specified exit pressure, and M_n is the face normal Mach number.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.28: Pressures at the Face of a Pressure Outlet Boundary



For incompressible flows, the face pressure is computed as an average between the specified pressure and the interior pressure.

$$P_f = 0.5\left(P_c + P_e\right) \tag{6.95}$$

In this boundary implementation, the exit pressure is not constant along the pressure outlet boundary. However, upon flow convergence, the average boundary pressure will be close to the specified static exit pressure.

6.3.9.4. Other Optional Inputs at Pressure Outlet Boundaries

6.3.9.4.1. Non-Reflecting Boundary Conditions Option

One of the options that may be used at pressure outlets is non-reflecting boundary conditions (NRBC). This option is used when waves are made to pass through the boundaries while avoiding false reflections. Details of non-reflecting boundary conditions can be found in General Non-Reflecting Boundary Conditions (p. 416).

6.3.9.4.2. Target Mass Flow Rate Option

The simple Bernoulli's equation is used to adjust the pressure at every iteration on a pressure outlet zone in order to meet the desired mass flow rate. The change in pressure, based on Bernoulli's equation is given by the following equation:

$$dP = 0.5\rho_{ave}\left(\dot{m}^2 - \dot{m}_{req}^2\right) / \left(\rho_{ave} A\right)^2 \tag{6.96}$$

where dP is the change in pressure, $\dot{m}$ is the current computed mass flow rate at the pressure-outlet boundary, $\dot{m}_{req}$ is the required mass flow rate, ρ_{ave} is the computed average density at the pressure-outlet boundary, and A is the area of the pressure-outlet boundary.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Limitations

- The target mass flow rate option is not available with multiphase flows or when any of the non-reflecting boundary conditions models are used.
- If the pressure-outlet zone is used in the mixing-plane model, the target mass flow rate option will not be available for that particular zone.
- The pressure outlet will not achieve the target mass flow rate if the flow becomes choked (that is, the Mach number of the fluid in the pressure-outlet zone becomes equal to 1).
- The pressure outlet may not achieve the target mass flow rate if porous zones are present in the flow domain causing a large pressure drop.

Target Mass Flow Rate Settings

To use the target mass flow rate option

1. Enable **Target Mass Flow Rate** in the **Pressure Outlet** dialog box.
2. Specify the **Target Mass Flow** as either a constant value or hook a UDF to set the target mass flow rate.

   The settings for the target mass flow rate option can be accessed from the `target-mass-flow-rate-settings` text command:

   `define` → `boundary-conditions` → `target-mass-flow-rate-settings`

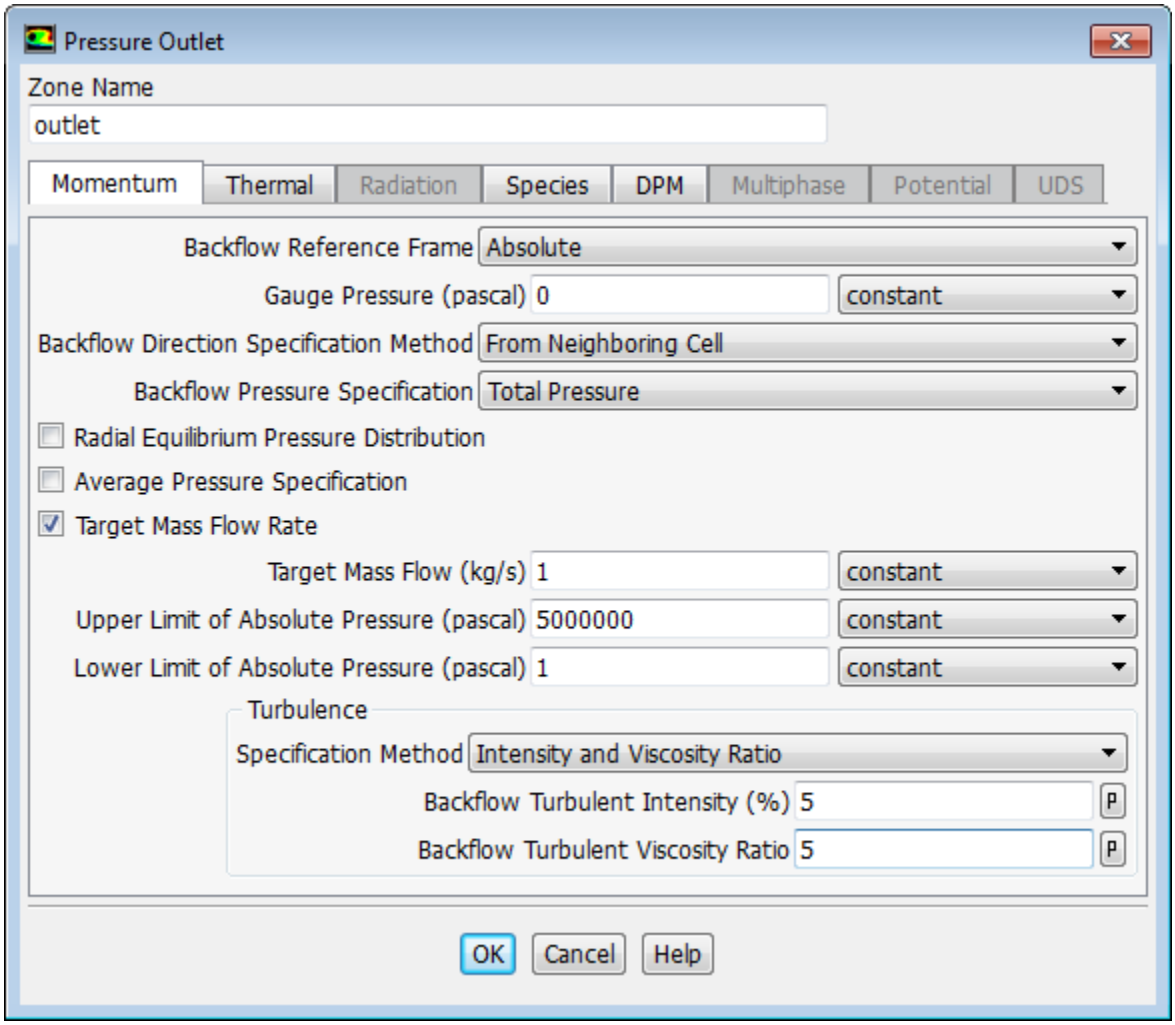
   You will be prompted to

   a. Set the under-relaxation factor (the default setting is 0.05).

   b. Enable the targeted mass flow rate verbosity (the default is `no`). If enabled, it prints to the console window the required mass flow rate, computed mass flow rate, mean pressure, the new pressure imposed on the outlet, and the change in pressure in SI units.
3. In the **Pressure Outlet** dialog box, specify the **Upper Limit of Absolute Pressure** and **Lower Limit of Absolute Pressure**. Specifying the range of the pressure limits improves convergence in cases with a large number of outlet boundaries, which have different pressure variations on different boundaries. You can also use the `define/boundary-conditions/pressure-outlet` text command to specify these limits.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.29: The Pressure Outlet Dialog Box with the Target Mass Flow Rate Option Enabled



Solution Strategies When Using the Target Mass Flow Rate Option

If convergence difficulties are encountered or if the solution is not converging at the desired mass flow rate, then try to lower the under-relaxation factor from the default value. Otherwise, you can use the alternate method to converge at the required mass flow rate.

In some cases, you may want to switch off the target mass flow rate option initially, then guess an exit pressure that will bring the solution closer to the target mass flow rate. After the solution stabilizes, you can turn on the target mass flow rate option and iterate to convergence. For many complex flow problems, this strategy is usually very successful.

The use of Full Multigrid Initialization is also very helpful in obtaining a good starting solution and in general will reduce the time required to get a converged solution on a target mass flow rate. For further information on Full Multigrid Initialization, see Full Multigrid (FMG) Initialization (p. 1595).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Target Mass Flow Rates Using UDFs

For some unsteady problems it is desirable that the target mass flow rate be a function of the physical flow time. This enforcement of boundary condition can be done by attaching a UDF with `DEFINE_PROFILE` functions to the target mass flow rate field.

Important

Note that the mass flow rate profile is a function of time and only one constant value should be applied to all zone faces at a given time.

An example of a simple UDF using a `DEFINE_PROFILE` that will adjust the mass flow rate can be found in `DEFINE_PROFILE` in the Fluent Customization Manual.

6.3.10. Pressure Far-Field Boundary Conditions

Pressure far-field conditions are used in ANSYS Fluent to model a free-stream condition at infinity, with free-stream Mach number and static conditions being specified. The pressure far-field boundary condition is often called a characteristic boundary condition, since it uses characteristic information (Riemann invariants) to determine the flow variables at the boundaries.

6.3.10.1. Limitations

Note the following limitations and restrictions when using pressure far-field boundary conditions:

- This boundary condition is applicable only when the density is calculated using the ideal-gas law (see Density (p. 479)). Using it for other flows is not permitted. To effectively approximate true infinite-extent conditions, you must place the far-field boundary far enough from the object of interest. For example, in lifting airfoil calculations, it is not uncommon for the far-field boundary to be a circle with a radius of 20 chord lengths.
- It is incompatible with the multiphase models (VOF, mixture, and Eulerian) that are available with the pressure-based solver.
- It cannot be applied to flows that employ constant density, the real gas model, and the wet steam model, which are available in the density-based solver.

For an overview of flow boundaries, see Flow Inlet and Exit Boundary Conditions (p. 302).

6.3.10.2. Inputs at Pressure Far-Field Boundaries

6.3.10.2.1. Summary

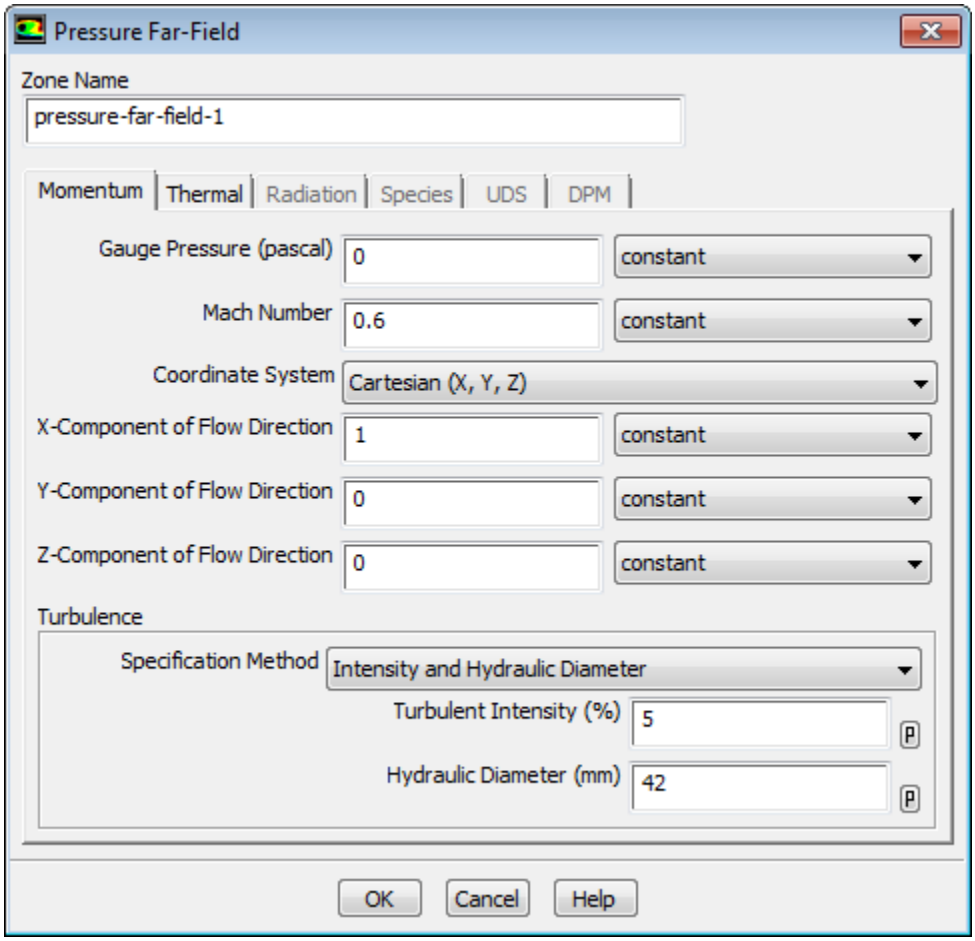
You will enter the following information for a pressure far-field boundary:

- static pressure
- Mach number
- temperature
- flow direction

- turbulence parameters (for turbulent calculations)
- radiation parameters (for calculations using the P-1, DTRM, DO, surface-to-surface, or MC models)
- chemical species mass or mole fractions (for species calculations)
- discrete phase boundary conditions (for discrete phase calculations)

All values are entered in the Pressure Far-Field Dialog Box (p. 2430) (Figure 6.30: The Pressure Far-Field Dialog Box (p. 348)), which is opened from the **Boundary Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 241)).

Figure 6.30: The Pressure Far-Field Dialog Box



6.3.10.2.2. Defining Static Pressure, Mach Number, and Static Temperature

To set the static pressure and temperature at the far-field boundary in the **Pressure Far-Field** dialog box, enter the appropriate values for **Gauge Pressure** and **Mach Number** in the **Momentum** tab. The Mach number can be subsonic, sonic, or supersonic. Set the **Temperature** in the **Thermal** tab.

6.3.10.2.3. Defining the Flow Direction

You can define the flow direction at a pressure far-field boundary by setting the components of the direction vector. If your geometry is 2D non-axisymmetric enter appropriate values for **X** and **Y** in the **Pressure Far-Field** dialog box (Figure 6.30: The Pressure Far-Field Dialog Box (p. 348)). If your geometry

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

is 2D axisymmetric, enter the appropriate values for **Axial**, **Radial**, and (if you are modeling axisymmetric swirl) **Tangential-Component of Flow Direction**.

If your geometry is 3D, you can choose a **Coordinate System** that is **Cartesian**, **Cylindrical**, or **Local Cylindrical**. In the Cartesian coordinate system, enter the appropriate values for **X**, **Y**, and **Z-Component of Flow Direction**. If the direction cosine data on the boundary is available, then use the cylindrical or local cylindrical coordinate system and specify the **Axial**, **Radial**, **Tangential-Component of Flow Direction**. For **Cylindrical**, axis parameters need to be specified on the adjacent cell zone of the boundary face. For **Local Cylindrical Swirl**, specify the **Axis Origin** and **Axis Direction**.

6.3.10.2.4. Defining Turbulence Parameters

For turbulent calculations, there are several ways in which you can define the turbulence parameters. Instructions for deciding which method to use and determining appropriate values for these inputs are provided in Determining Turbulence Parameters (p. 303). Turbulence modeling is described in Modeling Turbulence (p. 749).

6.3.10.2.5. Defining Radiation Parameters

If you are using the P-1, DTRM, DO, surface-to-surface, or MC models, you will set the **Internal Emissivity** and (optionally) **External Black Body Temperature Method**. See Defining Boundary Conditions for Radiation (p. 869) for details.

6.3.10.2.6. Defining Species Transport Parameters

If you are modeling species transport, you will set the species mass or mole fractions under **Species Mass Fractions** or **Species Mole Fractions**. See Defining Cell Zone and Boundary Conditions for Species (p. 990) for details.

6.3.10.3. Defining Discrete Phase Boundary Conditions

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the pressure far-field boundary. See Setting Boundary Conditions for the Discrete Phase (p. 1301) for details.

6.3.10.4. Default Settings at Pressure Far-Field Boundaries

Default settings (in SI) for pressure far-field boundary conditions are as follows:

Gauge Pressure	0
Mach Number	0.6
Temperature	300
X-Component of Flow Direction	1
Y-Component of Flow Direction	0
Z-Component of Flow Direction	0
Turbulent Intensity	5%
Turbulent Viscosity Ratio	10

6.3.10.5. Calculation Procedure at Pressure Far-Field Boundaries

The pressure far-field boundary condition is a non-reflecting boundary condition based on the introduction of Riemann invariants (that is, characteristic variables) for a one-dimensional flow normal to the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

boundary. For flow that is subsonic there are two Riemann invariants, corresponding to incoming and outgoing waves:

$$R_{\infty}=v_{n_{\infty}}-\frac{2c_{\infty}}{\gamma-1} \tag{6.97}$$

$$R_i=v_{n_i}+\frac{2c_i}{\gamma-1} \tag{6.98}$$

where v_n is the velocity magnitude normal to the boundary, c is the local speed of sound and γ is the ratio of specific heats (ideal gas). The subscript ∞ refers to conditions being applied at infinity (the boundary conditions), and the subscript i refers to conditions in the interior of the domain (that is, in the cell adjacent to the boundary face). These two invariants can be added and subtracted to give the following two equations:

$$v_n=\frac{1}{2}\left(R_i+R_{\infty}\right) \tag{6.99}$$

$$c=\frac{\gamma-1}{4}\left(R_i-R_{\infty}\right) \tag{6.100}$$

where v_n and c become the values of normal velocity and sound speed applied on the boundary. At a face through which flow exits, the tangential velocity components and entropy are extrapolated from the interior; at an inflow face, these are specified as having free-stream values. Using the values for v_n, c, tangential velocity components, and entropy the values of density, velocity, temperature, and pressure at the boundary face can be calculated.

6.3.11. Outflow Boundary Conditions

Outflow boundary conditions in ANSYS Fluent are used to model flow exits where the details of the flow velocity and pressure are not known prior to solving the flow problem. You do not define any conditions at outflow boundaries (unless you are modeling radiative heat transfer, a discrete phase of particles, or split mass flow): ANSYS Fluent extrapolates the required information from the interior. It is important, however, to understand the limitations of this boundary type.

Important

Note that outflow boundaries cannot be used in the following cases:

- If a problem includes pressure inlet boundaries; use pressure outlet boundary conditions (see Pressure Outlet Boundary Conditions (p. 338)) instead.
- If you are modeling compressible flow.
- If you are modeling unsteady flows with varying density (even if the fluid is incompressible), it is preferable to use a pressure outlet.

In general, an outflow condition may be used in incompressible cases using the Eulerian or Mixture multiphase models. However, if the flow may produce a recirculation at the outlet or if the flow field is not stable and fully developed at the outlet, then a pressure outlet boundary condition is preferred.

For an overview of flow boundaries, see Flow Inlet and Exit Boundary Conditions (p. 302).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.11.1. ANSYS Fluent's Treatment at Outflow Boundaries

The boundary conditions used by ANSYS Fluent at outflow boundaries are as follows:

- A zero diffusion flux for all flow variables.
- An overall mass balance correction.

The zero diffusion flux condition applied at outflow cells means that the conditions of the outflow plane are extrapolated from within the domain and have no impact on the upstream flow. The extrapolation procedure used by ANSYS Fluent updates the outflow velocity and pressure in a manner that is consistent with a fully-developed flow assumption, as noted below, when there is no area change at the outflow boundary.

The zero diffusion flux condition applied by ANSYS Fluent at outflow boundaries is approached physically in fully-developed flows. Fully-developed flows are flows in which the flow velocity profile (and/or profiles of other properties such as temperature) is unchanging in the flow direction.

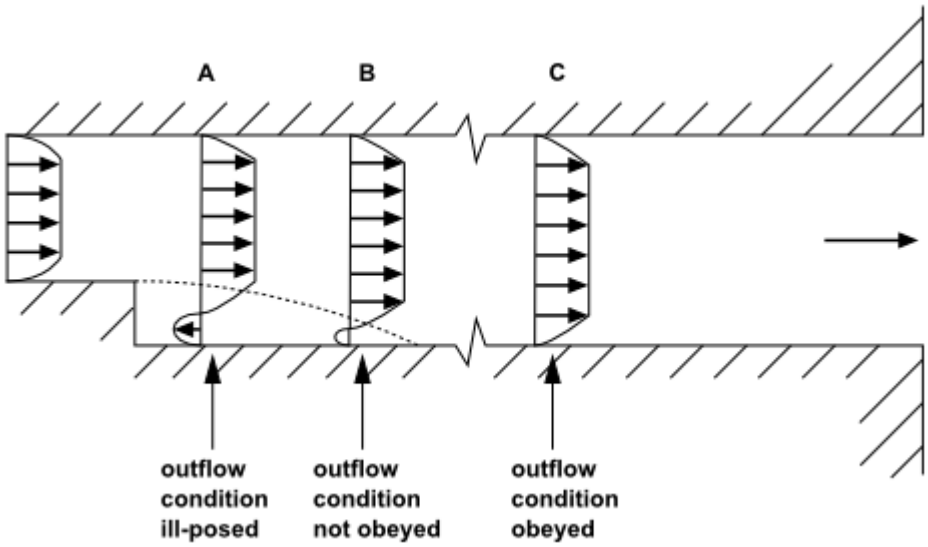
It is important to note that gradients in the cross-stream direction may exist at an outflow boundary. Only the diffusion fluxes in the direction normal to the exit plane are assumed to be zero.

6.3.11.2. Using Outflow Boundaries

As noted in ANSYS Fluent's Treatment at Outflow Boundaries (p. 351), the outflow boundary condition is obeyed in fully-developed flows where the diffusion flux for all flow variables in the exit direction are zero. However, you may also define outflow boundaries at physical boundaries where the flow is not fully developed—and you can do so with confidence if the assumption of a zero diffusion flux at the exit is expected to have a small impact on your flow solution. The appropriate placement of an outflow boundary is described by example below.

- Outflow boundaries where normal gradients are negligible: Figure 6.31: Choice of the Outflow Boundary Condition Location (p. 351) shows a simple two-dimensional flow problem and several possible outflow boundary location choices. Location C shows the outflow boundary located upstream of the plenum exit but in a region of the duct where the flow is fully-developed. At this location, the outflow boundary condition is exactly obeyed.

Figure 6.31: Choice of the Outflow Boundary Condition Location



- Ill-posed outflow boundaries: Location B in Figure 6.31: Choice of the Outflow Boundary Condition Location (p. 351) shows the outflow boundary near the reattachment point of the recirculation in the wake of the backward-facing step. This choice of outflow boundary condition is ill-posed as the gradients normal to the exit plane are quite large at this point and can be expected to have a significant impact on the flow field upstream. Because the outflow boundary condition ignores these axial gradients in the flow, location B is a poor choice for an outflow boundary. The exit location should be moved downstream from the reattachment point.

  Figure 6.31: Choice of the Outflow Boundary Condition Location (p. 351) shows a second ill-posed outflow boundary at location A. Here, the outflow is located where flow is pulled into the ANSYS Fluent domain through the outflow boundary. In situations like this the ANSYS Fluent calculation typically does not converge and the results of the calculation have no validity. This is because when flow is pulled into the domain through an outflow, the mass flow rate through the domain is "floating" or undefined. In addition, when flow enters the domain through an outflow boundary, the scalar properties of the flow are not defined. For example, the temperature of the flow pulled in through the outflow is not defined. (ANSYS Fluent chooses the temperature using the temperature of the fluid adjacent to the outflow, inside the domain.) Therefore you should view all calculations that involve flow entering the domain through an outflow boundary with skepticism. For such calculations, pressure outlet boundary conditions (see Pressure Outlet Boundary Conditions (p. 338)) are recommended.

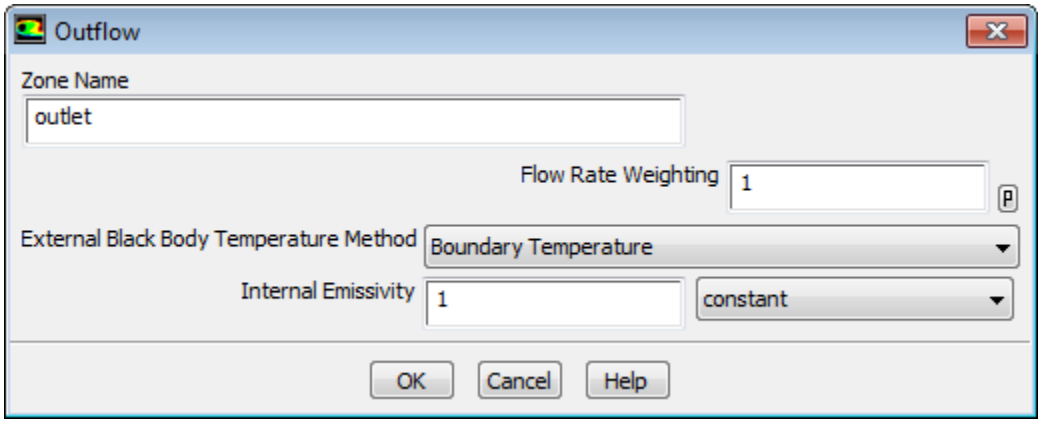
Important

Note that convergence may be affected if there is recirculation through the outflow boundary at any point during the calculation, even if the final solution is not expected to have any flow reentering the domain. This is particularly true of turbulent flow simulations.

6.3.11.3. Mass Flow Split Boundary Conditions

In ANSYS Fluent, it is possible to use multiple outflow boundaries and specify the fractional flow rate through each boundary. In the Outflow Dialog Box (p. 2420), set the **Flow Rate Weighting** to indicate what portion of the outflow is through the boundary.

Figure 6.32: The Outflow Dialog Box



The **Flow Rate Weighting** is a weighting factor:

$$\text{percentage flow through boundary} = \frac{\text{Flow Rate Weighting specified on boundary}}{\text{sum of all Flow Rate Weightings}} \tag{6.101}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

By default, the **Flow Rate Weighting** for all outflow boundaries is set to 1. If the flow is divided equally among all of your outflow boundaries (or if you have just one outflow boundary), you need not change the settings from the default; ANSYS Fluent will scale the flow rate fractions to obtain equal fractions through all outflow boundaries. Therefore, if you have two outflow boundaries and you want half of the flow to exit through each one, no inputs are required from you. If, however, you want 75% of the flow to exit through one, and 25% through the other, you will need to explicitly specify *both* **Flow Rate Weighting** values, that is, 0.75 for one boundary and 0.25 for the other.

Important

If you specify a **Flow Rate Weighting** of 0.75 at the first exit and leave the default **Flow Rate Weighting** (1.0) at the second exit, then the flow through each boundary will be

$$\text{Boundary 1} = \frac{0.75}{0.75+1.0} = 0.429 \text{ or } 42.9\%$$

$$\text{Boundary 2} = \frac{1.0}{0.75+1.0} = 0.571 \text{ or } 57.1\%$$

6.3.11.4. Other Inputs at Outflow Boundaries

6.3.11.4.1. Radiation Inputs at Outflow Boundaries

In general, there are no boundary conditions for you to set at an outflow boundary. If, however, you are using the P-1, DTRM, DO, surface-to-surface, or MC models, you will set the **Internal Emissivity** and (optionally) **External Black Body Temperature Method** in the **Outflow** dialog box. These parameters are described in Defining Boundary Conditions for Radiation (p. 869). The default value for **Internal Emissivity** is 1 and the default value for **Black Body Temperature** is 300.

6.3.11.4.2. Defining Discrete Phase Boundary Conditions

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the outflow boundary. See Setting Boundary Conditions for the Discrete Phase (p. 1301) for details.

6.3.12. Outlet Vent Boundary Conditions

Outlet vent boundary conditions are used to model an outlet vent with a specified loss coefficient and ambient (discharge) pressure and temperature.

6.3.12.1. Inputs at Outlet Vent Boundaries

You will enter the following information for an outlet vent boundary:

- static pressure
- backflow conditions
  - total (stagnation) temperature (for energy calculations)
  - backflow pressure specification
  - backflow direction specification method
  - turbulence parameters (for turbulent calculations)

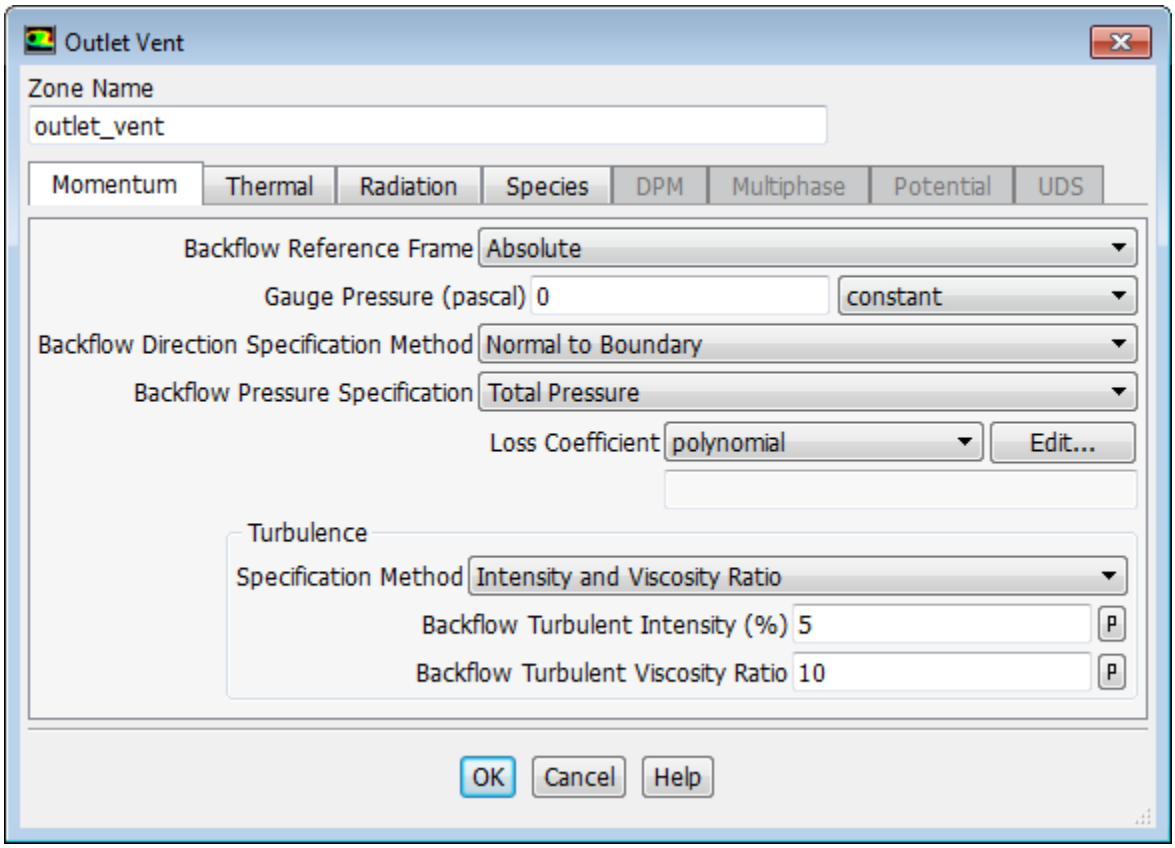
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - chemical species mass or mole fractions (for species calculations)
  - mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
  - progress variable (for premixed or partially premixed combustion calculations)
  - multiphase boundary conditions (for general multiphase calculations)
  - type of backflow reference frame (not available for multiphase flows)
- radiation parameters (for calculations using the P-1, DTRM, DO, surface-to-surface, or MC models)
- discrete phase boundary conditions (for discrete phase calculations)
- loss coefficient
- open channel flow parameters (for open channel flow calculations using the VOF multiphase model)

All values are entered in the Outlet Vent Dialog Box (p. 2422) (Figure 6.33: The Outlet Vent Dialog Box (p. 354)), which is opened from the **Boundary Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 241)).

The first 4 items listed above are specified in the same way that they are specified at pressure outlet boundaries. See Inputs at Pressure Outlet Boundaries (p. 338) for details. Specification of the loss coefficient is described here. Open channel boundary condition inputs are described in Modeling Open Channel Flows (p. 1410).

Figure 6.33: The Outlet Vent Dialog Box



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.12.1.1. Specifying the Loss Coefficient

An outlet vent is considered to be infinitely thin, and the pressure drop through the vent is assumed to be proportional to the dynamic head of the fluid, with an empirically determined loss coefficient that you supply. That is, the pressure drop, Δp, varies with the normal component of velocity through the vent, v, as follows:

$$\Delta p = k_L \frac{1}{2} \rho v^2 \tag{6.102}$$

where ρ is the fluid density, and k_L is the nondimensional loss coefficient.

Important

Δp is the pressure drop in the direction of the flow; therefore the vent will appear as a resistance even in the case of backflow.

You can define a **constant**, **polynomial**, **piecewise-linear**, or **piecewise-polynomial** function for the **Loss Coefficient** across the vent. The dialog boxes for defining these functions are the same as those used for defining temperature-dependent properties. See Defining Properties Using Temperature-Dependent Functions (p. 475) for details.

6.3.13. Exhaust Fan Boundary Conditions

Exhaust fan boundary conditions are used to model an external exhaust fan with a specified pressure jump and ambient (discharge) pressure.

Note

The exhaust fan boundary condition considers only momentum. Diffusion processes are not considered.

6.3.13.1. Inputs at Exhaust Fan Boundaries

You will enter the following information for an exhaust fan boundary:

- static pressure
- backflow conditions
  - total (stagnation) temperature (for energy calculations)
  - backflow pressure specification
  - backflow direction specification method
  - turbulence parameters (for turbulent calculations)
  - chemical species mass or mole fractions (for species calculations)
  - mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
  - progress variable (for premixed or partially premixed combustion calculations)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - multiphase boundary conditions (for general multiphase calculations)
  - type of backflow reference frame (not available for multiphase flows)
  - user-defined scalar boundary conditions (for user-defined scalar calculations)
- radiation parameters (for calculations using the P-1, DTRM, DO, surface-to-surface, or MC models)
- discrete phase boundary conditions (for discrete phase calculations)
- pressure jump
- open channel flow parameters (for open channel flow calculations using the VOF multiphase model)

All values are entered in the Exhaust Fan Dialog Box (p. 2393) (Figure 6.34: The Exhaust Fan Dialog Box (p. 357)), which is opened from the **Boundary Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 241)).

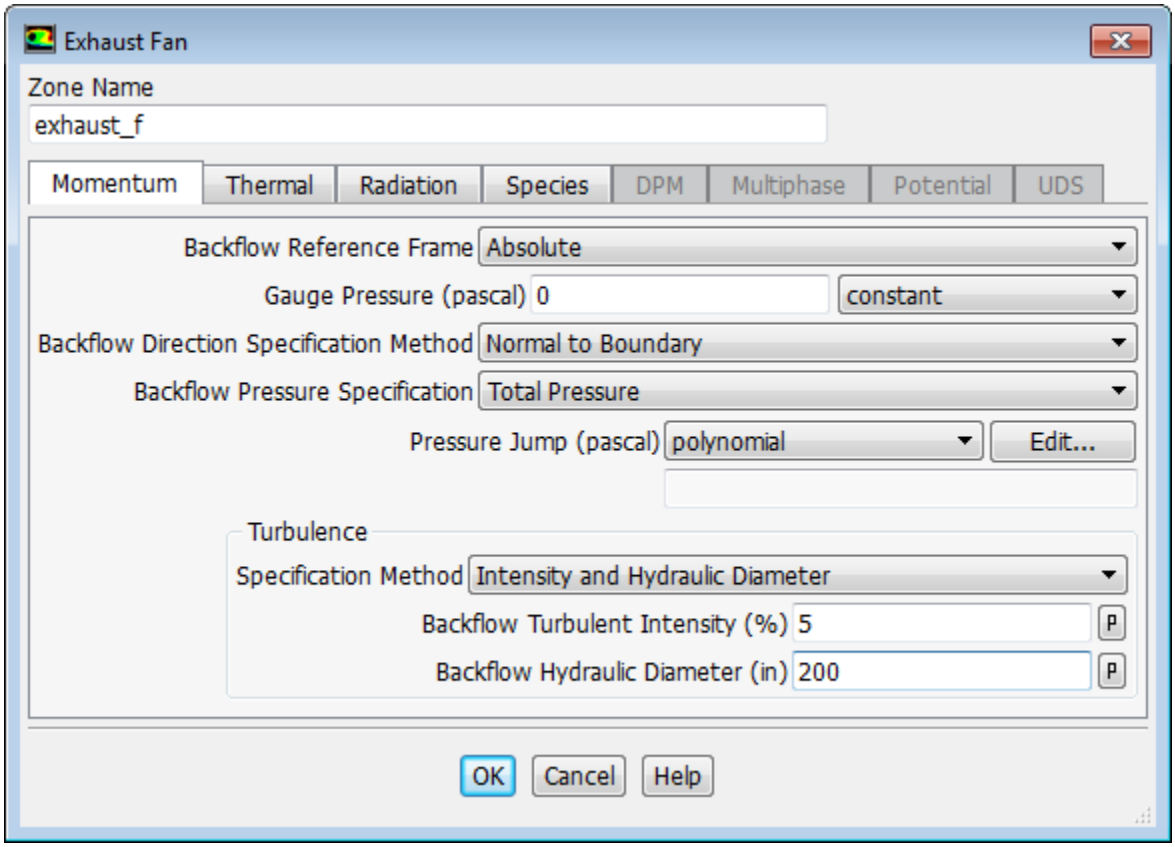
The first 4 items listed above are specified in the same way that they are specified at pressure outlet boundaries. See Inputs at Pressure Outlet Boundaries (p. 338) for details. Specification of the pressure jump is described here. Open channel boundary condition inputs are described in Modeling Open Channel Flows (p. 1410).

6.3.13.1.1. Specifying the Pressure Jump

An exhaust fan is considered to be infinitely thin, and the discontinuous pressure rise across it is specified as a function of the local fluid velocity normal to the fan. You can define a **constant**, **polynomial**, **piecewise-linear**, or **piecewise-polynomial** function for the **Pressure Jump** across the fan. The dialog boxes for defining these functions are the same as those used for defining temperature-dependent properties. See Defining Properties Using Temperature-Dependent Functions (p. 475) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.34: The Exhaust Fan Dialog Box



Important

You must be careful to model the exhaust fan so that a pressure rise occurs for forward flow through the fan. In the case of reversed flow, the fan is treated like an inlet vent with a loss coefficient of unity.

6.3.14. Degassing Boundary Conditions

Degassing boundary conditions are used to model a free surface through which dispersed gas bubbles are allowed to escape, but the continuous phase is not. A typical application is a bubble column in which you want to reduce computational cost by not including the freeboard region in the simulation.

When the degassing boundary condition is specified for an outlet, the continuous liquid phase sees the boundary as a free-slip wall and does not leave the domain. The dispersed gas phase sees the boundary as an outlet. The outlet pressure is not specified. Instead, ANSYS Fluent automatically specifies a mass sink for the dispersed gas phase in the cells adjacent to the degassing outlet. The mass sink is calculated using the flux normal to the boundary at the cell center.

Note

The degassing boundary condition is only available for two-phase liquid-gas flows using the Eulerian multiphase model.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.14.1. Limitations

The following limitations apply to the degassing boundary condition in ANSYS Fluent:

- The degassing boundary condition is only available for liquid-gas two-phase flow using the Eulerian model. The primary phase must be liquid.
- In order for the gas to escape from the degassing boundary, gravity must be switched on in the model.
- The degassing boundary condition is only recommended for modeling situations like bubble columns without the freeboard region.
- When postprocessing on a degassing boundary, there is no normal velocity for either phase since the gas escape is modeled by a mass sink in the neighboring cells.

6.3.14.2. Inputs at Degassing Boundaries

No inputs are necessary for the degassing boundary condition. However, the initial condition for volume fraction must be set appropriately. It is recommended that the gas phase volume fraction be initialized with a nonzero value smaller than the steady-state gas holdup value.

6.3.15. Wall Boundary Conditions

Wall boundary conditions are used to bound fluid and solid regions. In viscous flows, the no-slip boundary condition is enforced at walls by default, but you can specify a tangential velocity component in terms of the translational or rotational motion of the wall boundary, or model a "slip" wall by specifying shear. (You can also model a slip wall with zero shear using the symmetry boundary type, but using a symmetry boundary will apply symmetry conditions for *all* equations. See Symmetry Boundary Conditions (p. 384) for details.)

The shear stress and heat transfer between the fluid and wall are computed based on the flow details in the local flow field.

6.3.15.1. Inputs at Wall Boundaries

6.3.15.1.1. Summary

You will enter the following information for a wall boundary:

- wall motion conditions (for moving or rotating walls)
- shear conditions (for slip walls, optional)
- wall roughness (for turbulent flows, optional)
- thermal boundary conditions (for heat transfer calculations)
- species boundary conditions (for species calculations)
- chemical reaction boundary conditions (for surface reactions)
- radiation boundary conditions (for calculations using the P-1, DTRM, DO, surface-to-surface, or MC models)
- discrete phase boundary conditions (for discrete phase calculations)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

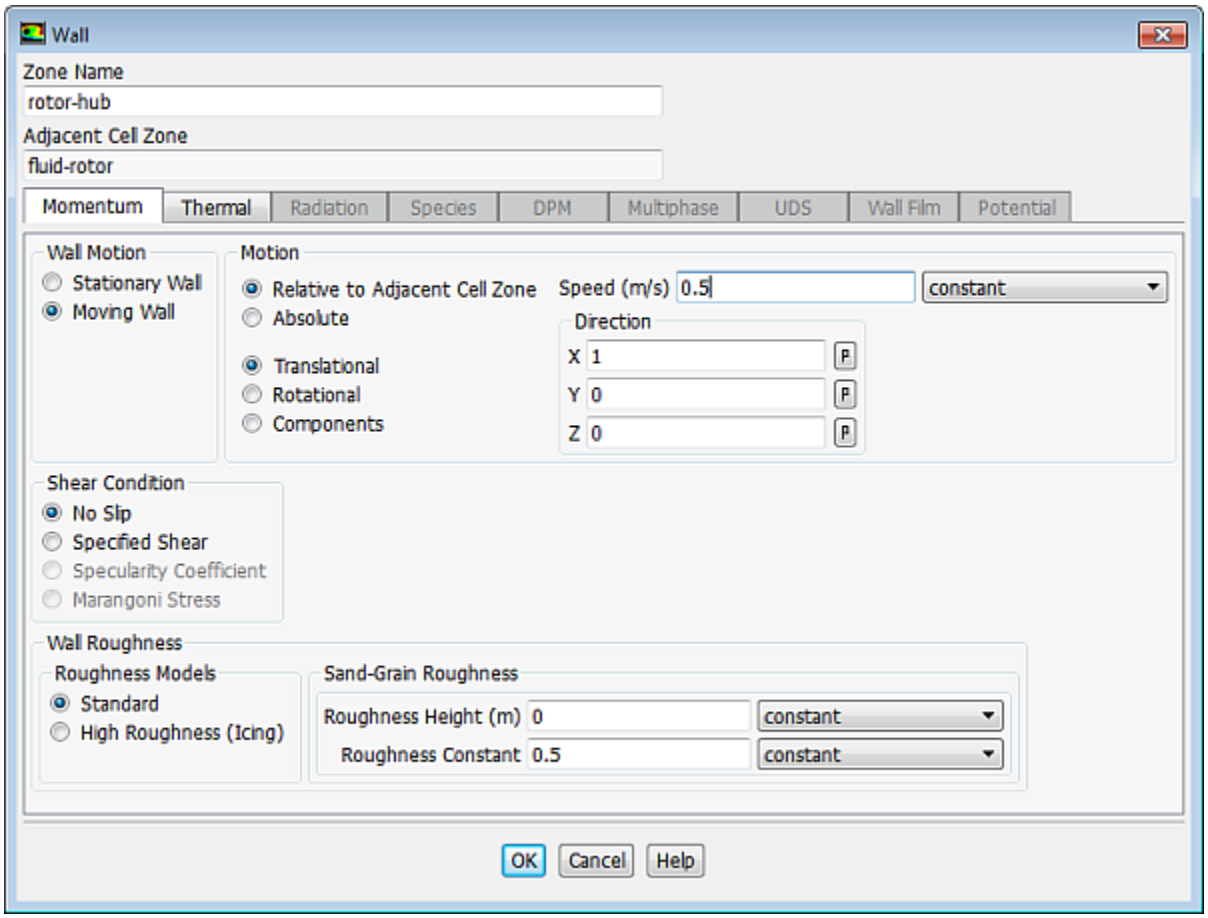
- wall adhesion contact angle (for VOF calculations, optional)

6.3.15.2. Wall Motion

Wall boundaries can be either stationary or moving. The stationary boundary condition specifies a fixed wall, whereas the moving boundary condition can be used to specify the translational or rotational velocity of the wall, or the velocity components.

Wall motion conditions are entered in the **Momentum** tab of the Wall Dialog Box (p. 2456) (Figure 6.35: The Wall Dialog Box for a Moving Wall (p. 359)), which is opened from the Boundary Conditions Task Page (p. 2389) (as described in Setting Cell Zone and Boundary Conditions (p. 241)). To view the wall motion conditions, click the **Momentum** tab.

Figure 6.35: The Wall Dialog Box for a Moving Wall



6.3.15.2.1. Defining a Stationary Wall

For a stationary wall, choose the **Stationary Wall** option under **Wall Motion**.

6.3.15.2.2. Velocity Conditions for Moving Walls

If you want to include tangential motion of the wall in your calculation, you need to define the translational or rotational velocity, or the velocity components. Select the **Moving Wall** option under **Wall Motion**. The **Wall** dialog box will expand, as shown in Figure 6.35: The Wall Dialog Box for a Moving Wall (p. 359), to show the wall velocity conditions.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that you cannot use the moving wall condition to model problems where the wall motion with respect to the adjacent cell zone has a component that is normal to the wall itself. For such problems, consider using a Sliding or Dynamic Mesh approach as discussed in Modeling Flows Using Sliding and Dynamic Meshes (p. 627). ANSYS Fluent will neglect any normal component of wall motion that you specify using the methods below.

- **Specifying Relative or Absolute Velocity**

  If the cell zone adjacent to the wall is moving (for example, if you are using a moving reference frame or a sliding mesh), you can choose to specify velocities relative to the zone motion by enabling the **Relative to Adjacent Cell Zone** option. If you choose to specify relative velocities, a velocity of zero means that the wall is stationary in the relative frame, and therefore moving at the speed of the adjacent cell zone in the absolute frame. If you choose to specify absolute velocities (by enabling the **Absolute** option), a velocity of zero means that the wall is stationary in the absolute frame, and therefore moving at the speed of the adjacent cell zone—but in the opposite direction—in the relative reference frame.

  Important

  If you are using one or more moving reference frames, sliding meshes, or mixing planes, and you want the wall to be fixed in the moving frame, it is recommended that you specify relative velocities (the default) rather than absolute velocities. Then, if you modify the speed of the adjacent cell zone, you will not need to make any changes to the wall velocities, as you would if you specified absolute velocities.

  Note that if the adjacent cell zone is not moving, the absolute and relative options are equivalent.

- **Translational Wall Motion**

  For problems that include linear translational motion of the wall boundary (for example, a rectangular duct with a moving belt as one wall) you can enable the **Translational** option and specify the wall's **Speed** and **Direction** (**X,Y,Z** vector). By default, wall motion is "disabled" by the specification of **Translational** velocity with a **Speed** of zero. You can define time-varying translational speed using a transient profile (Standard Transient Profiles (p. 449)) or user-defined function for the speed of the wall.

  If you need to define nonlinear translational motion, you will need to use the **Components** option, described below.

- **Rotational Wall Motion**

  For problems that include rotational wall motion you can enable the **Rotational** option and define the rotational **Speed** about a specified axis. To define the axis, set the **Rotation-Axis Direction** and **Rotation-Axis Origin**. This axis is independent of the axis of rotation used by the adjacent cell zone, and independent of any other wall rotation axis. For 3D problems, the axis of rotation is the vector passing through the specified **Rotation-Axis Origin** and parallel to the vector from (0,0,0) to the (**X,Y,Z**) point specified under **Rotation-Axis Direction**. For 2D problems, you will specify only the **Rotation-Axis Origin**; the axis of rotation is the z-direction vector passing through the specified point. For 2D axisymmetric problems, you will not define the axis: the rotation will always be about the x axis, with the origin at (0,0). You can define time-varying rotational speed using a transient profile (Standard Transient Profiles (p. 449)) profile or a user-defined function for the speed of the wall.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that the modeling of tangential rotational motion will be correct only if the wall bounds a surface of revolution about the prescribed axis of rotation (for example, a circle or cylinder). Note also that rotational motion can be specified for a wall in a stationary reference frame.

- **Wall Motion Based on Velocity Components**

  For problems that include linear or nonlinear translational motion of the wall boundary you can enable the **Components** option and specify the **X-Velocity**, **Y-Velocity**, and **Z-Velocity** of the wall. You can define nonlinear translational motion using a profile or a user-defined function for the **X-Velocity**, **Y-Velocity**, and/or **Z-Velocity** of the wall.

- **Wall Motion for Two-Sided Walls**

  As discussed earlier in this section, when you read a mesh with a two-sided wall zone (which forms the interface between fluid/solid regions) into ANSYS Fluent, a "shadow" zone will automatically be created so that each side of the wall is a distinct wall zone. For two-sided walls, it is possible to specify different motions for the wall and shadow zones, whether or not they are coupled. Note, however, that you cannot specify motion for a wall (or shadow) that is adjacent to a solid zone.

6.3.15.2.3. Shear Conditions at Walls

Four types of shear conditions are available:

- no-slip
- specified shear
- specularity coefficient
- Marangoni stress

The no-slip condition is the default, and it indicates that the fluid sticks to the wall and moves with the same velocity as the wall, if it is moving. The specified shear and Marangoni stress boundary conditions are useful in modeling situations in which the shear stress (rather than the motion of the fluid) is known. Examples of such situations are applied shear stress, slip wall (zero shear stress), and free surface conditions (zero shear stress or shear stress dependent on surface tension gradient). The specified shear boundary condition allows you to specify the x, y, and z components of the shear stress as constant values or profiles. The Marangoni stress boundary condition allows you to specify the gradient of the surface tension with respect to the temperature at this surface. The shear stress is calculated based on the surface gradient of the temperature and the specified surface tension gradient. The Marangoni stress option is available only for calculations in which the energy equation is being solved.

The specularity coefficient shear condition is specifically used in multiphase with granular flows. The specularity coefficient is a measure of the fraction of collisions that transfer momentum to the wall and its value ranges between zero and unity. This implementation is based on the Johnson and Jackson [48] (p. 2889) boundary conditions for granular flows.

Shear conditions are entered in the **Momentum** tab of the Wall Dialog Box (p. 2456), which is opened from the Boundary Conditions Task Page (p. 2389) (as described in Setting Cell Zone and Boundary Conditions (p. 241)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

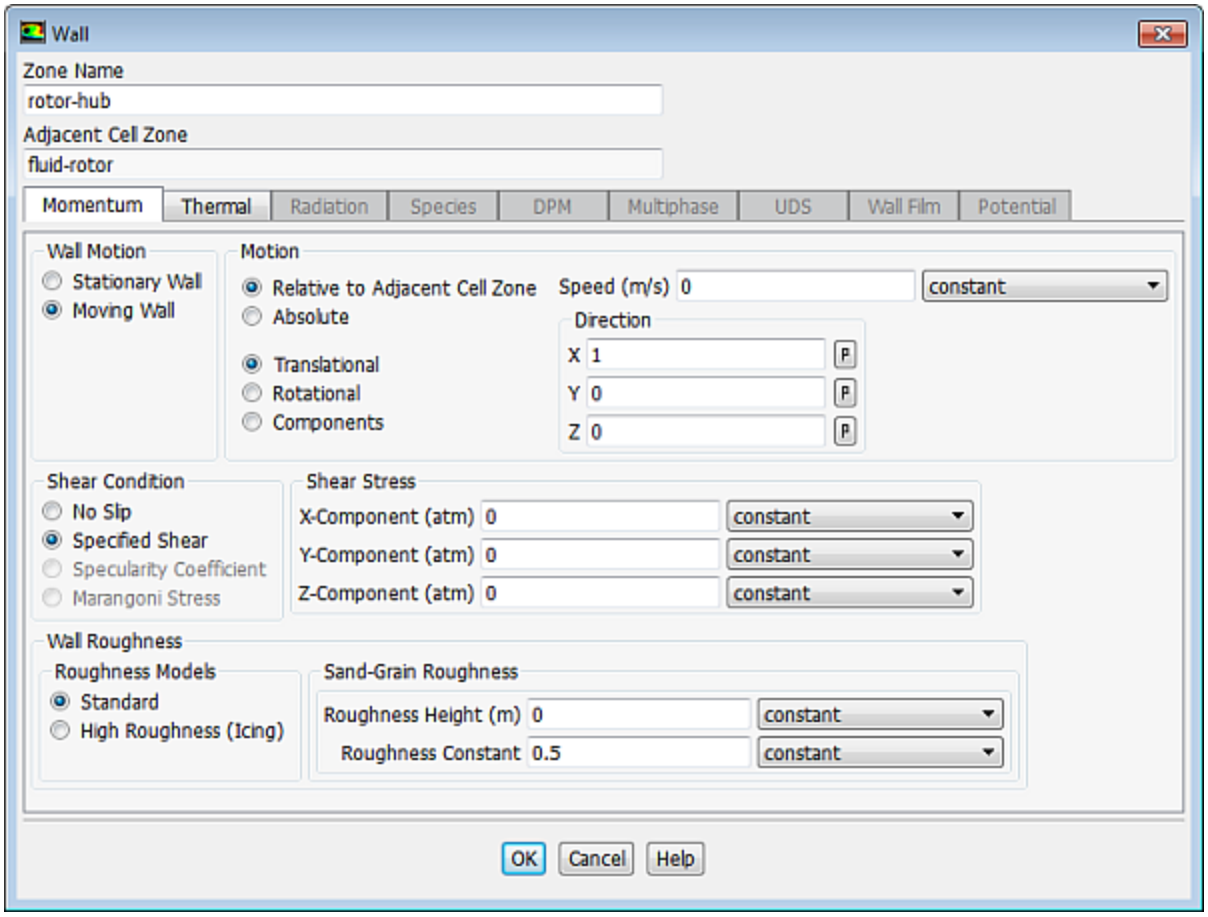
6.3.15.2.4. No-Slip Walls

You can model a no-slip wall by selecting the **No Slip** option under **Shear Condition**. This is the default for all walls in viscous flows.

6.3.15.2.5. Specified Shear

In addition to the no-slip wall that is the default for viscous flows, you can model a slip wall by specifying zero or nonzero shear. For nonzero shear, the shear to be specified is the shear at the wall by the fluid. To specify the shear, select the **Specified Shear** option under **Shear Condition** (see Figure 6.36: The Wall Dialog Box for Specified Shear (p. 362)). You can then enter x, y, and z components of shear under **Shear Stress**. Wall functions for turbulence are not used with the **Specified Shear** option.

Figure 6.36: The Wall Dialog Box for Specified Shear



6.3.15.2.6. Specularity Coefficient

For multiphase granular flow, you can specify the specularity coefficient such that when the value is zero, this condition is equivalent to zero shear at the wall, but when the value is near unity, there is a significant amount of lateral momentum transfer. To specify the specularity coefficient, select the **Specularity Coefficient** option under **Shear Condition** (see Figure 6.37: The Wall Dialog Box for the

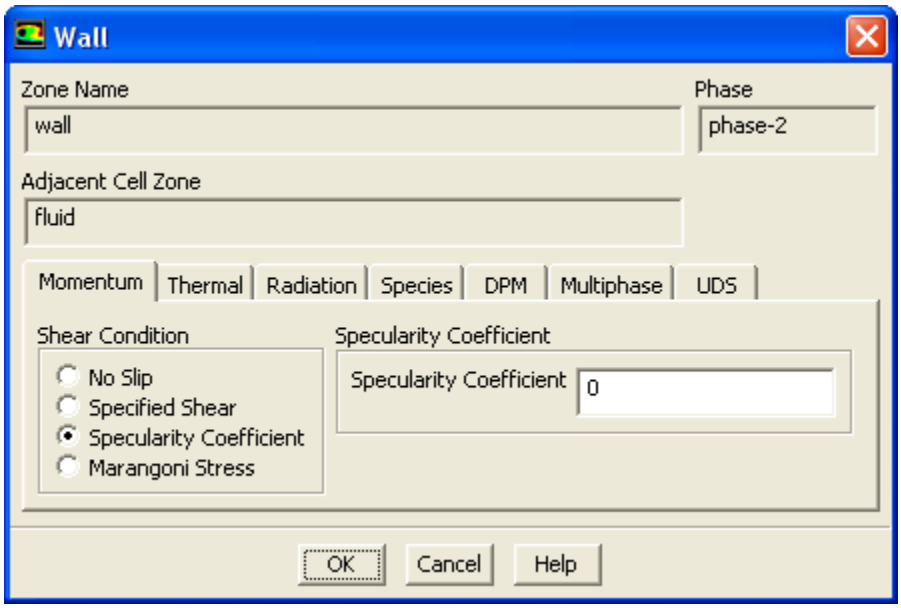
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specularity Coefficient (p. 363)) and enter the desired value in the text-entry box under **Specularity Coefficient**.

Note

The specularity coefficient is not available for moving walls.

Figure 6.37: The Wall Dialog Box for the Specularity Coefficient



6.3.15.2.7. Marangoni Stress

ANSYS Fluent can also model shear stresses caused by the variation of surface tension due to temperature. The shear stress applied at the wall is given by

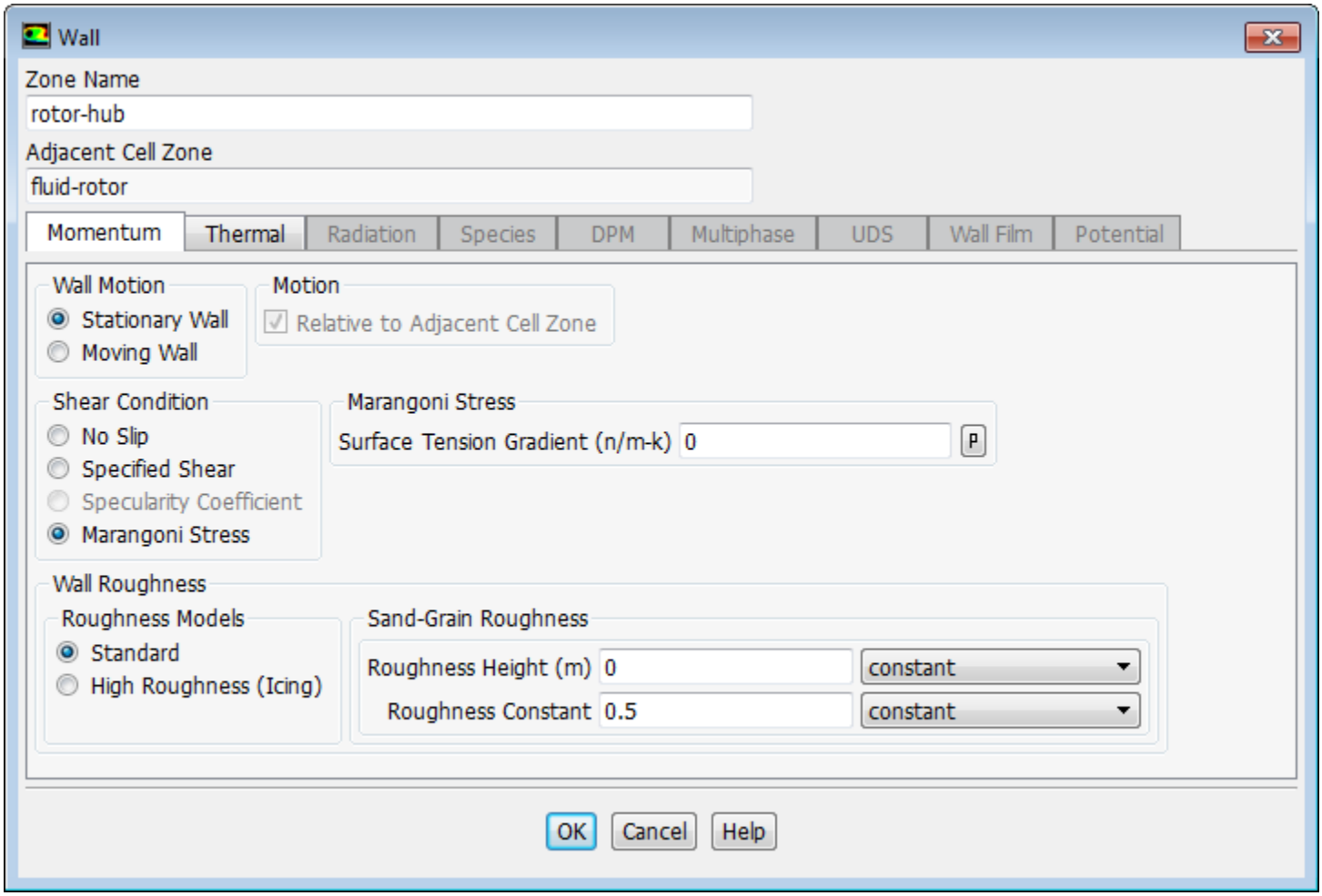
$$\tau = \frac{d\sigma}{dT} \nabla_s T \tag{6.103}$$

where $d\sigma / dT$ is the surface tension gradient with respect to temperature, and $\nabla_s T$ is the surface gradient. This shear stress is then applied to the momentum equation.

To model Marangoni stress for the wall, select the **Marangoni Stress** option under **Shear Condition** (see Figure 6.38: The Wall Dialog Box for Marangoni Stress (p. 364)). This option is available only for calculations in which the energy equation is being solved. You can then enter the surface tension gradient ($d\sigma / dT$ in Equation 6.103 (p. 363)) in the **Surface Tension Gradient** field. Wall functions for turbulence are not used with the **Marangoni Stress** option.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.38: The Wall Dialog Box for Marangoni Stress



6.3.15.2.8. Wall Roughness Effects in Turbulent Wall-Bounded Flows

Fluid flows over rough surfaces are encountered in diverse situations. Examples are, among many others, flows over the surfaces of airplanes (especially due to ice accretion), ships, turbomachinery, heat exchangers, and piping systems, and atmospheric boundary layers over terrain of varying roughness. Wall roughness affects drag (resistance) and heat and mass transfer on the walls.

If you are modeling a turbulent wall-bounded flow in which the wall roughness effects are considered to be significant, you can include the wall roughness effects through the Standard Law-of-the-Wall Modified for Roughness (p. 364) law-of-the-wall modified for roughness or you can use one of the Additional Roughness Models for Icing Simulations (p. 368).

6.3.15.2.8.1. Standard Law-of-the-Wall Modified for Roughness

Experiments in roughened pipes and channels indicate that the mean velocity distribution near rough walls, when plotted in the usual semi-logarithmic scale, has the same slope ($1/\kappa$) but a different intercept (additive constant B in the log-law). Therefore, the law-of-the-wall for mean velocity modified for roughness has the form

$$\frac{u_p u^*}{\tau_w/\rho} = \frac{1}{\kappa}\ln\left(E\frac{\rho u^* y_p}{\mu}\right) - \Delta B \tag{6.104}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where $u^*=C_\mu^{1/4}k^{1/2}$ and

$$\Delta B=\frac{1}{\kappa}\ln f_r \tag{6.105}$$

where f_r is a roughness function that quantifies the shift of the intercept due to roughness effects.

ΔB depends, in general, on the type (uniform sand, rivets, threads, ribs, mesh-wire, and so on) and size of the roughness. There is no universal roughness function valid for all types of roughness. For a sand-grain roughness and similar types of uniform roughness elements, however, ΔB has been found to be well-correlated with the nondimensional roughness height, $K_s^+=\rho K_s u^*/\mu$, where K_s is the physical roughness height and $u^*=C_\mu^{1/4}k^{1/2}$. Analyses of experimental data show that the roughness function is not a single function of K_s^+, but takes different forms depending on the K_s^+ value. It has been observed that there are three distinct regimes:

- hydrodynamically smooth ($K_s^+\le 2.25$)
- transitional ($2.25<K_s^+\le 90$)
- fully rough ($K_s^+>90$)

According to the data, roughness effects are negligible in the hydrodynamically smooth regime, but become increasingly important in the transitional regime, and take full effect in the fully rough regime.

In ANSYS Fluent, the whole roughness regime is subdivided into the three regimes, and the formulas proposed by Cebeci and Bradshaw based on Nikuradse's data [20] (p. 2888) are adopted to compute ΔB for each regime.

For the hydrodynamically smooth regime ($K_s^+\le 2.25$):

$$\Delta B=0 \tag{6.106}$$

For the transitional regime ($2.25<K_s^+\le 90$):

$$\Delta B=\frac{1}{\kappa}\ln\left[\frac{K_s^+-2.25}{87.75}+C_s K_s^+\right]\times\sin\left\{0.4258\left(\ln K_s^+-0.811\right)\right\} \tag{6.107}$$

where C_s is a roughness constant, and depends on the type of the roughness.

In the fully rough regime ($K_s^+>90$):

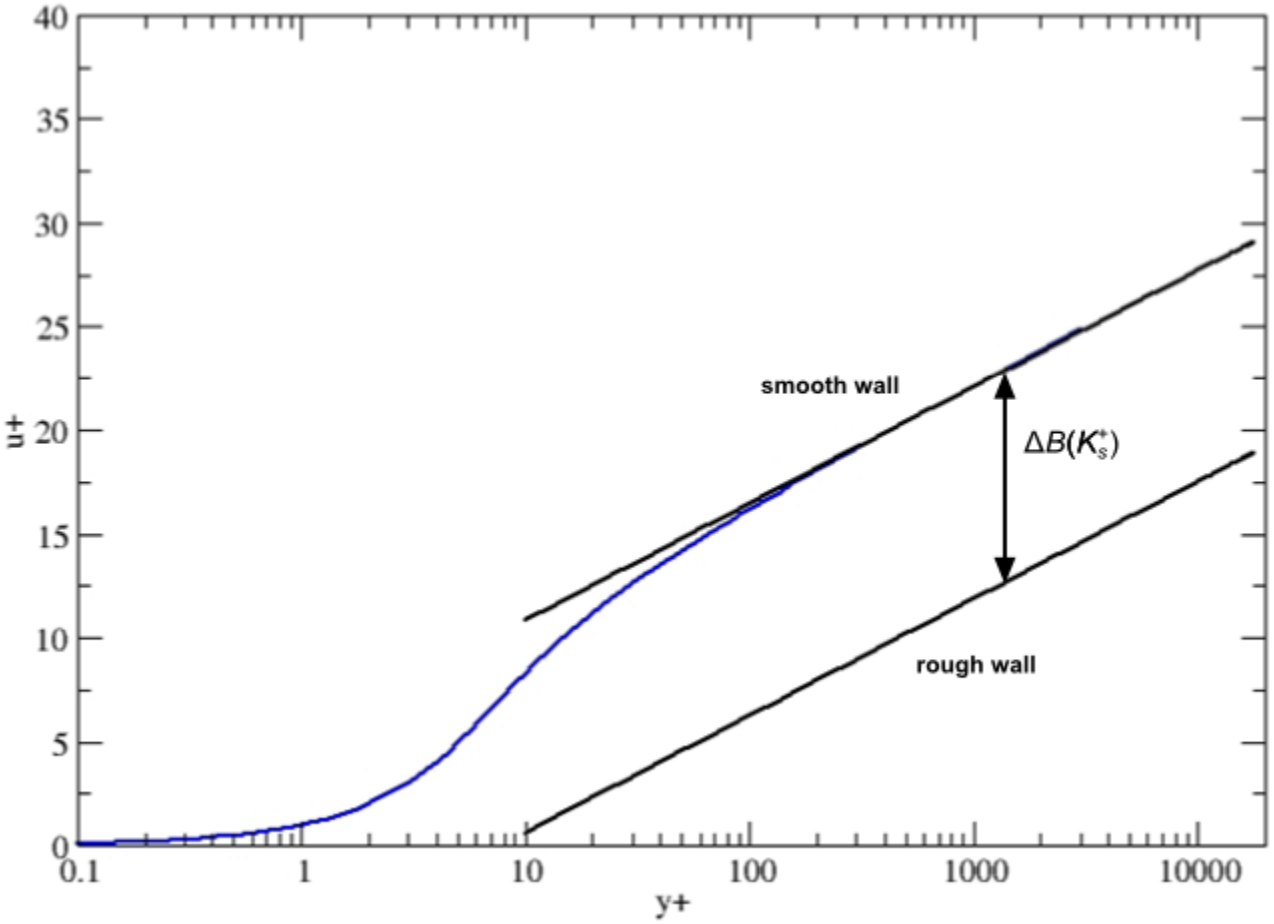
$$\Delta B=\frac{1}{\kappa}\ln\left(1+C_s K_s^+\right) \tag{6.108}$$

In the solver, given the roughness parameters, $\Delta B\left(K_s^+\right)$ is evaluated using the corresponding formula (Equation 6.106 (p. 365), Equation 6.107 (p. 365), or Equation 6.108 (p. 365)). The modified law-of-the-wall in Equation 6.104 (p. 364) is then used to evaluate the shear stress at the wall and other wall functions for the mean temperature and turbulent quantities.

$\Delta B\left(K_s^+\right)$ represents a downward shift of the logarithmic velocity profile, as shown in the following figure:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.39: Downward Shift of the Logarithmic Velocity Profile



This downward shift leads to a singularity for large roughness heights and low values of y^+. Depending on the turbulence model and near wall treatment, two different approaches are used in ANSYS Fluent in order to avoid this issue:

- reducing the roughness height as y^+ decreases

  The first approach consists in redefining the roughness height based on the mesh refinement:

  $$K_s^+ = \min\left(K_s^+, y^+\right) \tag{6.109}$$

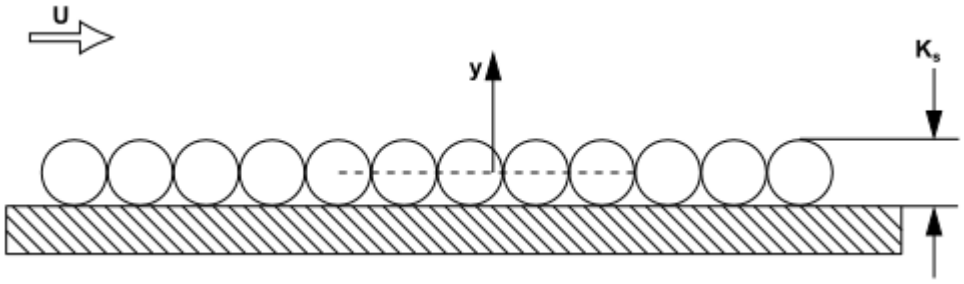
  This ensures that as y^+ approaches zero, so too does K_s^+. Therefore, the mesh requirement for rough walls in this case is $y^+ > K_s^+$, in order to maintain the full effect of the roughness on the flow.

- virtually shifting the wall

  The second approach is based on the observation that the viscous sublayer is fully established only near hydraulically smooth walls. In the transitional roughness regime, the roughness elements are slightly thicker than the viscous sublayer and start to disturb it, so that in fully rough flows, the sublayer is destroyed and viscous effects become negligible. The following figure illustrates the equivalent sand-grain roughness using a wall with a layer of closely packed spheres, which have an average roughness height representing a technical roughness with peaks and valleys of different shapes and sizes (see Schlichting and Gersten [101] (p. 2892)):

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.40: Illustration of Equivalent Sand-Grain Roughness



It can be assumed that the roughness has a blockage effect, which is about 50% of its height (note that the figure above shows a two-dimensional cut of a three-dimensional arrangement).

It is therefore sensible to virtually shift the wall to 50% of the height of the roughness elements. This results in a corrected y^+ value for the first cell center:

$$y^+ = y^+ + K_s^+ / 2 \tag{6.110}$$

which gives about the correct displacement caused by the surface roughness. Thus the singularity issue is avoided and fine meshes can be handled correctly.

The second approach (that is, virtually shifting the wall) is the default treatment for rough walls for all two-equation turbulence models based on the ω-equation and for the following turbulence models based on the ε-equation, when they are used with standard and scalable wall functions (note that the use of scalable wall functions is recommended over the use of standard wall functions):

- standard, RNG, and realizable k-ε model
- Reynolds stress models

All other model combinations with rough walls (for example, the Spalart-Allmaras model) have no special calibration on fine meshes, and therefore the first approach (reducing the roughness height as y^+ decreases) is used.

Note

Rough walls cannot be used together with the following model combinations:

- an ε-equation model with enhanced wall treatment or the Menter-Lechner near-wall treatment

  Note that the following are the relevant ε-equation models:

  – all of the k-ε models (that is, standard, RNG, and realizable)

  – the Reynolds stress model with the **Linear Pressure-Strain** model selected

  – the detached eddy simulation (DES) model with the **Realizable k-epsilon** option selected

- the Reynolds stress model with the **Stress-Omega** or **Stress-BSL** model selected
- the transition k-kl-ω model

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- the large eddy simulation (LES) model

Important

Prior to ANSYS Fluent 14, the shift described by Equation 6.110 (p. 367) was not applied when using turbulence models based on the ε-equation. You can recover the previous code behavior by using the following scheme command:

```
(rpsetvar 'ke-rough-wall-treatment-r14? #f)
(models-changed)
```

6.3.15.2.8.1.1. Setting the Roughness Parameters

The roughness parameters are in the **Momentum** tab of the Wall Dialog Box (p. 2456) (see Figure 6.38: The Wall Dialog Box for Marangoni Stress (p. 364)), which is opened from the Boundary Conditions Task Page (p. 2389) (as described in Setting Cell Zone and Boundary Conditions (p. 241)).

To model the wall roughness effects, you must specify two roughness parameters: the **Roughness Height**, K_s, and the **Roughness Constant**, C_s. The default roughness height (K_s) is zero, which corresponds to smooth walls. For the roughness to take effect, you must specify a nonzero value for K_s. For a uniform sand-grain roughness, the height of the sand-grain can simply be taken for K_s. For a non-uniform sand-grain, however, the mean diameter (D_{50}) would be a more meaningful roughness height. For other types of roughness, an "equivalent" sand-grain roughness height could be used for K_s. The above approaches are only relevant if the height is considered constant per surface. However, if the roughness constant or roughness height is not constant, then you can specify a profile (see Profiles (p. 435)). Similarly, user-defined functions may be used to define a wall roughness height that is not constant. For details on the format of user-defined functions, refer to the Fluent Customization Manual.

Choosing a proper roughness constant (C_s) is dictated mainly by the type of the given roughness. The default roughness constant (C_s=0.5) was determined so that, when used with k- ε turbulence models, it reproduces Nikuradse's resistance data for pipes roughened with tightly-packed, uniform sand-grain roughness. You may need to adjust the roughness constant when the roughness you want to model departs much from uniform sand-grain. For instance, there is some experimental evidence that, for non-uniform sand-grains, ribs, and wire-mesh roughness, a higher value (C_s=0.5∼1.0) is more appropriate. Unfortunately, a clear guideline for choosing C_s for arbitrary types of roughness is not available.

Note

The advantage of the rough wall formulation using a virtual shift of the wall (Equation 6.110 (p. 367)) compared to reducing the roughness height as y^+ decreases (Equation 6.109 (p. 366)) is that it eliminates all restrictions with respect to mesh resolution near the wall, and can therefore be used on arbitrarily fine meshes.

6.3.15.2.8.2. Additional Roughness Models for Icing Simulations

Additional roughness models are available if you have selected the **Spalart-Allmaras** or the **SST** k-ω model in the **Viscous Model** dialog box. See the Steps in Using a Turbulence Model (p. 764) for more information. With one of these viscous models active, the additional roughness models will be available on the **Wall** boundary condition panel on the **Momentum** tab under **Wall Roughness** → **Roughness**

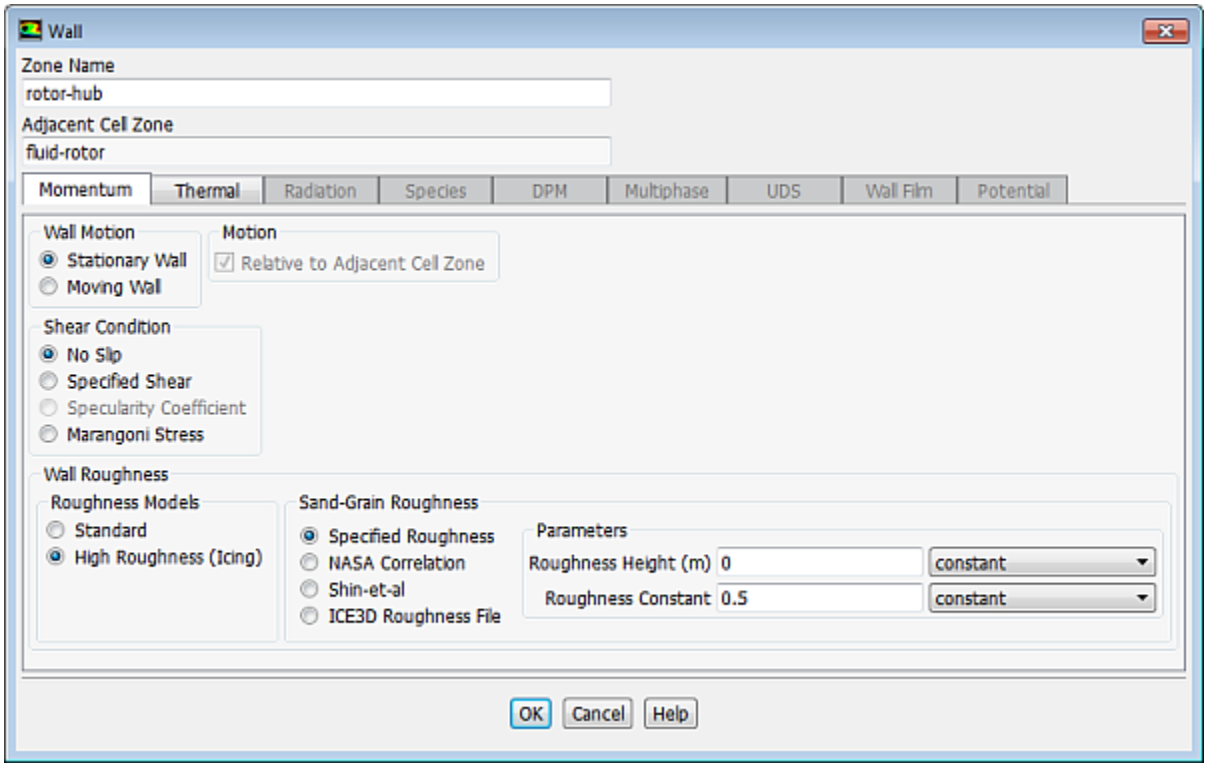
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Models → **High Roughness (Icing)**. A discussion of the treatment of the turbulence models with the **High Roughness (Icing)** enabled is given in Treatment of the Spalart-Allmaras Model for Icing Simulations and Treatment of the SST Model for Icing Simulations. These models are primarily designed and tested for simulations of icing applications, however they may also be useful for other applications in which the boundary layer is fully resolved and there is surface roughness that is large relative to the near-wall mesh.

The additional roughness models have the following limitations:

- When using the **High Roughness (Icing)** models and the **Spalart-Allmaras** model, you can only run the solver in parallel. You can read a case in serial for set up and postprocessing, but you cannot run the solver in serial.
- The **High Roughness (Icing)** models are only valid for low-Re number turbulence, or need fine near-wall meshes (mesh should fully resolve the boundary layer).

Figure 6.41: The Wall Dialog Box forHigh Roughness (Icing) Models



You have the following options:

- Specified Roughness
- NASA Correlation
- Shin-et-al
- ICE3D Roughness File

6.3.15.2.8.2.1. Specified Roughness

The **Specified Roughness** option is the same as in the **Standard** roughness model. See Setting the Roughness Parameters (p. 368) for more details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.15.2.8.2.2. NASA Correlation

If the **NASA Correlation** option is selected, the surface sand-grain roughness height is computed with an empirical NASA correlation for icing in air flows [103] (p. 2892). The sand-grain roughness height is computed from the product of the following coefficients:

$$\left[\frac{h_s/c}{\left(h_s/c\right)_{base}}\right]_{V_\infty} = 0.4286 + 0.0044139 V_\infty \tag{6.111}$$

$$\left[\frac{h_s/c}{\left(h_s/c\right)_{base}}\right]_{LC} = 0.5714 + 0.2457\,(LC) + 1.2571\,(LC)^2 \tag{6.112}$$

$$\left[\frac{h_s/c}{\left(h_s/c\right)_{base}}\right]_{T_s} = 0.04\,T_s - 11.27 \tag{6.113}$$

Where V_∞ is the **Free Stream Velocity**, T_s is the **Free Stream Temperature**, LC is the **Liquid Content** (in most cases water for ice accretion), c is the **Characteristic Length**, and $\left(h_s/c\right)_{base} = 0.001177$.

The sand-grain roughness height is then obtained from the formulation:

$$h_s = \left[\frac{h_s/c}{\left(h_s/c\right)_{base}}\right]_{T_s}\left[\frac{h_s/c}{\left(h_s/c\right)_{base}}\right]_{LC}\left[\frac{h_s/c}{\left(h_s/c\right)_{base}}\right]_{V_\infty} \cdot \left(h_s/c\right)_{base} \cdot c \tag{6.114}$$

6.3.15.2.8.2.3. Shin-et-al

If the **Shin-et-al** option is selected the empirical correlation for the surface sand-grain roughness is computed with the Shin and Bond formulation [103] (p. 2892), which modifies the NASA correlation (see the NASA Correlation (p. 370) for a description of variables) with the following factor:

$$\left[\frac{h_s/c}{\left(h_s/c\right)_{base}}\right]_{MVD} = \begin{cases} 1 & MVD \le 20 \\ 1.667 - 0.0333(MVD) & MVD > 20 \end{cases} \tag{6.115}$$

where MVD is the droplet mean diameter (**Droplet Diameter**). The corresponding value of the sand-grain roughness height is obtained from:

$$h_s = 0.6839\left[\frac{h_s/c}{\left(h_s/c\right)_{base}}\right]_{LC}\left[\frac{h_s/c}{\left(h_s/c\right)_{base}}\right]_{T_s}\left[\frac{h_s/c}{\left(h_s/c\right)_{base}}\right]_{MVD} \cdot \left(h_s/c\right)_{base} \cdot c \tag{6.116}$$

6.3.15.2.8.2.4. ICE3D Roughness File

If the **ICE3D Roughness File** is selected, the sand-grain roughness height is read from a node-based input file obtained from FENSAP-ICE. The file must have the format $\{x_i, y_i, z_i, h_{s,i}\}$, and be read in Fluent via **File** → **Read** → **Profile**. After the FENSAP-ICE roughness file is read into Fluent, the file name will appear in the dropdown list.

6.3.15.3. Thermal Boundary Conditions at Walls

When you are solving the energy equation, you need to define thermal boundary conditions at wall boundaries. Seven types of thermal conditions are available:

- fixed heat flux

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- fixed temperature
- convective heat transfer
- external radiation heat transfer
- combined external radiation and convection heat transfer
- thermal data transferred between another system in Workbench using System Coupling
- thermal coupling across a mapped interface

If the wall zone is a "two-sided wall" (a wall that forms the interface between two regions, such as the fluid/solid interface for a conjugate heat transfer problem) a subset of these thermal conditions will be available, but you will also be able to choose whether or not the two sides of the wall are "coupled". See below for details.

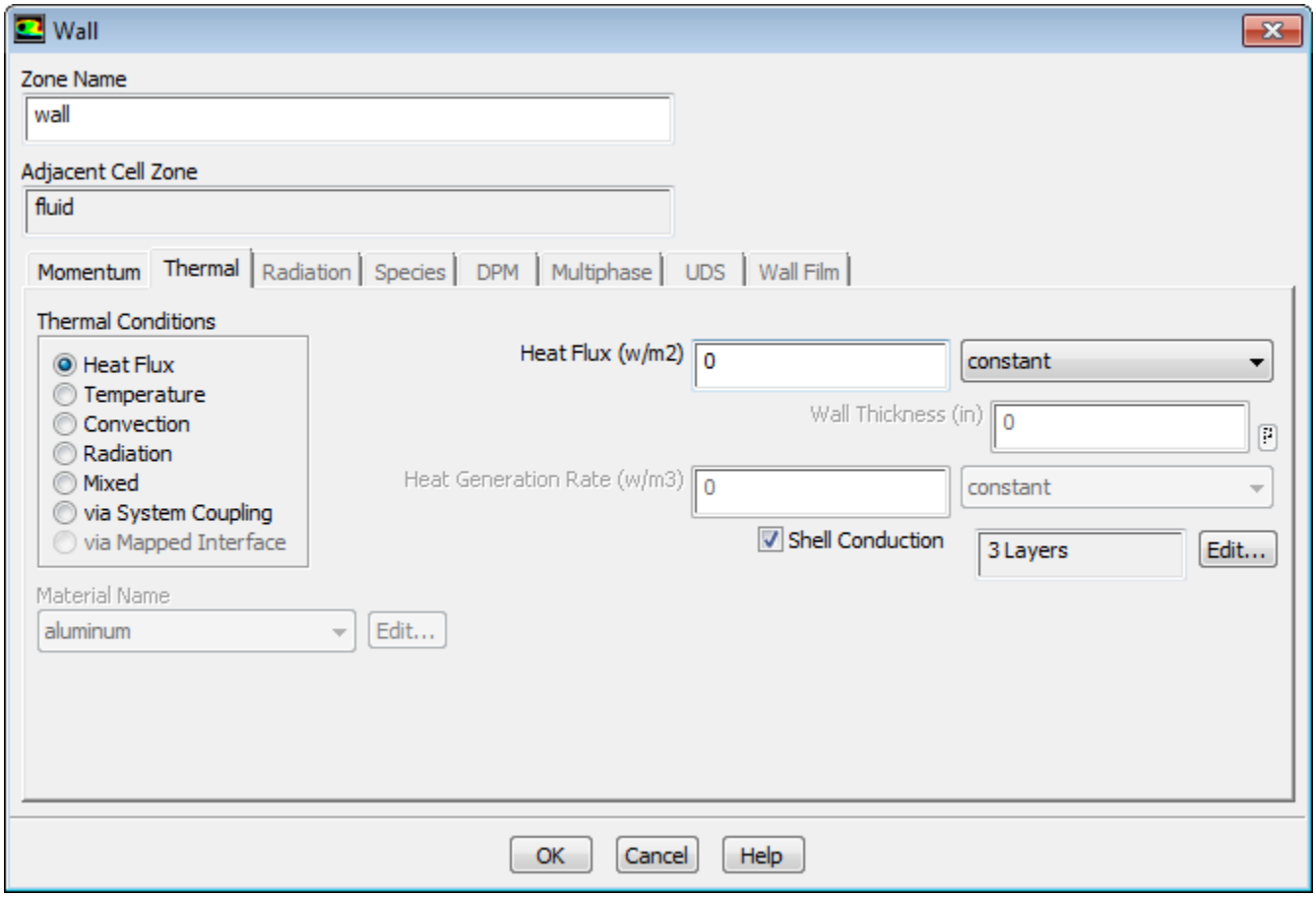
The inputs for each type of thermal condition are described below. If the wall has a nonzero-thickness, you should also set parameters for calculating thin-wall thermal resistance and heat generation in the wall, as described below.

You can model conduction within boundary walls and internal (that is, two-sided) walls of your model. This type of conduction, called shell conduction, allows you to more conveniently model heat conduction on walls where the wall thickness is small with respect to the overall geometry (for example, finned heat exchangers or sheet metal in automobile underhoods). Meshing these walls with solid cells would lead to high-aspect-ratio meshes and a significant increase in the total number of cells. See below for details about shell conduction.

Thermal conditions are entered in the **Thermal** tab of the Wall Dialog Box (p. 2456) (Figure 6.42: The Wall Dialog Box (Thermal Tab) (p. 372)), which is opened from the **Boundary Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 241)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.42: The Wall Dialog Box (Thermal Tab)



6.3.15.3.1. Heat Flux Boundary Conditions

For a fixed heat flux condition, choose the **Heat Flux** option under **Thermal Conditions**. You will then need to set the appropriate value for the heat flux at the wall surface in the **Heat Flux** field. You can define an adiabatic wall by setting a zero heat flux condition. This is the default condition for all walls.

6.3.15.3.2. Temperature Boundary Conditions

To select the fixed temperature condition, choose the **Temperature** option under **Thermal Conditions** in the **Wall** dialog box. You will need to specify the temperature at the wall surface (**Temperature**). The heat transfer to the wall is computed using Equation 6.118 (p. 382) or Equation 6.119 (p. 382).

6.3.15.3.3. Convective Heat Transfer Boundary Conditions

For a convective heat transfer wall boundary, select **Convection** under **Thermal Conditions**. Your inputs of **Heat Transfer Coefficient** and **Free Stream Temperature** will allow ANSYS Fluent to compute the heat transfer to the wall using Equation 6.122 (p. 383).

6.3.15.3.4. External Radiation Boundary Conditions

If radiation heat transfer from the exterior of your model is of interest, you can enable the **Radiation** option in the **Wall** dialog box and set the **External Emissivity** and **External Radiation Temperature**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.15.3.5. Combined Convection and External Radiation Boundary Conditions

You can choose a thermal condition that combines the convection and radiation boundary conditions by selecting the **Mixed** option. With this thermal condition, you will need to set the **Heat Transfer Coefficient**, **Free Stream Temperature**, **External Emissivity**, and **External Radiation Temperature**.

6.3.15.3.6. Augmented Heat Transfer

When modeling the heat transfer of applications that have perturbed flow and/or disturbed boundary layers, it can be necessary to augment the calculation of the diffusive heat flux with a convective augmentation factor. Such applications include the modeling of underhood and underbody heat loads, as wells as transient heat transfer in fully warmed-up exhaust systems.

The convective augmentation factor represents the ratio of the measured Nusselt number to the Nusselt number of an ideal flow. You can define it by using the following text command:

`define` → `boundary-conditions` → `wall`

You will be prompted to define the `Convective Augmentation Factor` as either a profile or a single value. Note that a value of 1 (the default value) represents no augmentation of the diffusive heat flux, whereas values greater than 1 initiate augmentation. For further details, see the equation for `qid` in `DEFINE_HEAT_FLUX` of the Fluent Customization Manual.

6.3.15.3.7. Thin-Wall Thermal Resistance Parameters

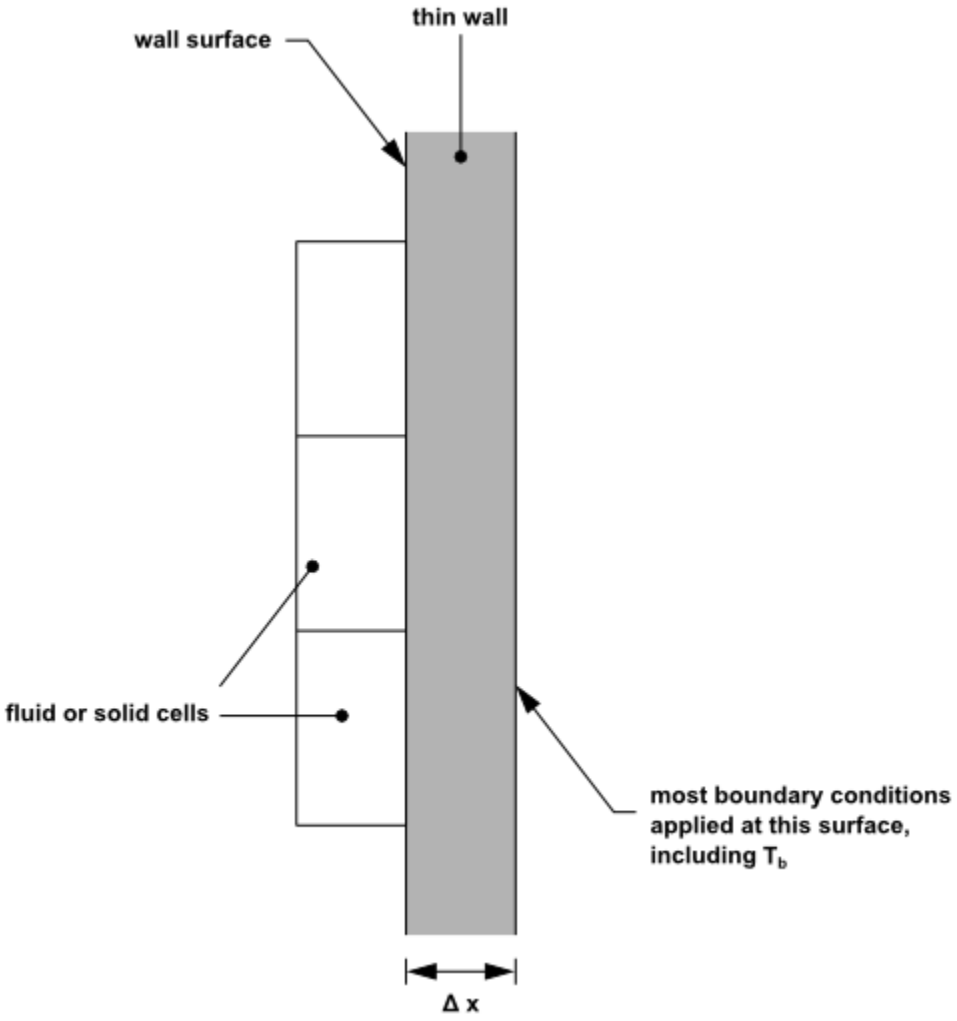
By default, a wall will have a thickness of zero. You can, however, in conjunction with any of the thermal conditions, model a thin layer of material on the wall. For example, you can model the effect of a piece of sheet metal between two fluid zones, a coating on a solid zone, or contact resistance between two solid regions. ANSYS Fluent will solve a 1D steady heat conduction equation to compute the thermal resistance offered by the wall and the heat generation in the wall.

To include these effects in the heat transfer calculation you will need to specify the type of material, the thickness of the wall, and the heat generation rate in the wall. Select the material type in the **Material Name** drop-down list, and specify the thickness in the **Wall Thickness** field. If you want to check or modify the properties of the selected material, you can click **Edit...** to open the **Edit Material** dialog box; this dialog box contains just the properties of the selected material, not the full contents of the standard **Create/Edit Materials** dialog box.

When you specify a thickness, the wall is then treated as a coupled wall, where the surface that is adjacent to the fluid / solid cells is referred to as the "wall surface". See Figure 6.43: A Thin Wall (p. 374).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.43: A Thin Wall



The thermal resistance of the wall is $\Delta x / k$, where k is the conductivity of the wall material and Δx is the wall thickness. The thermal wall boundary condition you set will be specified on the surface that is separated from the fluid / solid cells by the wall thickness. The temperature specified at this side of the wall is T_b.

Important

Note that for thin walls, you can only specify a constant thermal conductivity. If you want to use a non-constant thermal conductivity for a wall with nonzero-thickness, you should use the shell conduction model (see Shell Conduction (p. 376) for details).

Specify the heat generation rate inside the wall in the **Heat Generation Rate** field. This option is useful if, for example, you are modeling printed circuit boards where you know the electrical power dissipated in the circuits.

When postprocessing a wall that has a thickness but does not have shell conduction enabled, the **Temperature...** category provides three options: the temperature of the adjacent fluid / solid cells are stored as **Static Temperature**; the temperature of the wall surface itself is stored as **Wall Temperature**; and the temperature of the surface that is separated from the fluid / solid cells by the wall thickness is

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

stored as **Wall Temperature (Thin)**. If a more detailed analysis of the solid zone and surfaces is required, then you should consider creating layers of solid cells in your meshing application.

6.3.15.3.8. Thermal Conditions for Two-Sided Walls

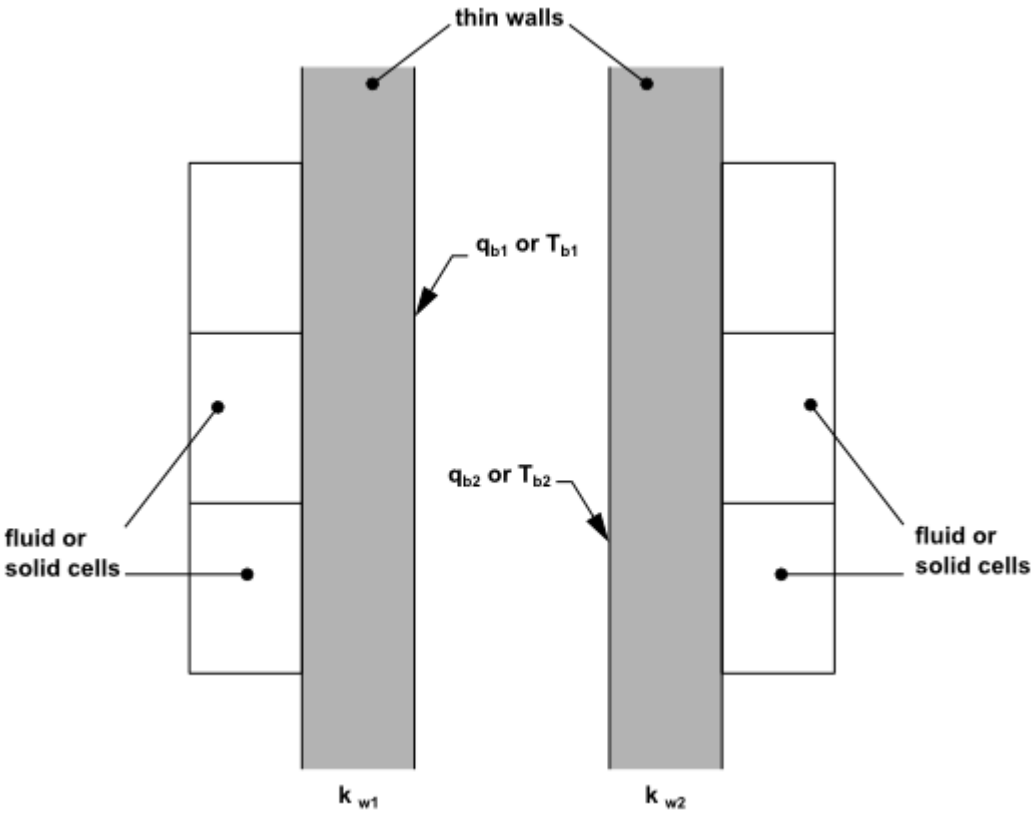
If the wall zone has a fluid or solid region on each side, it is called a "two-sided wall". When you read a mesh with this type of wall zone into ANSYS Fluent, a "shadow" zone will automatically be created so that each side of the wall is a distinct wall zone. In the **Wall** dialog box, the shadow zone's name will be shown in the **Shadow Face Zone** field. You can choose to specify different thermal conditions on each zone, or to couple the two zones:

- To couple the two sides of the wall, select the **Coupled** option under **Thermal Conditions**. (This option will appear in the **Wall** dialog box only when the wall is a two-sided wall.) No additional thermal boundary conditions are required, because the solver will calculate heat transfer directly from the solution in the adjacent cells. You can, however, specify the material type, wall thickness, and heat generation rate for thin-wall thermal resistance calculations, as described above. Note that the resistance parameters you set for one side of the wall will automatically be assigned to its shadow wall zone. Specifying the heat generation rate inside the wall is useful if, for example, you are modeling printed circuit boards where you know the electrical power dissipated in the circuits but not the heat flux or wall temperature.

- To uncouple the two sides of the wall and specify different thermal conditions on each one, choose **Temperature** or **Heat Flux** as the thermal condition type (**Convection** and **Radiation** are not applicable for two-sided walls); note that this uncoupling will not be effective if you have enabled shell conduction for the wall. The relationship between the wall and its shadow will be retained, so that you can couple them again at a later time, if desired. You will need to set the relevant parameters for the selected thermal condition, as described above. The two uncoupled walls can have different thicknesses, and are effectively insulated from one another. If you specify a nonzero wall thickness for the uncoupled walls, the thermal boundary conditions you set will be specified for each thin wall on the surface that is separated from the fluid / solid cells by the wall thickness, as shown in Figure 6.44: Uncoupled Thin Walls (p. 376), where T_{b1} is the **Temperature** (or q_{b1} is the **Heat Flux**) specified on one wall and T_{b2} is the **Temperature** (or q_{b2} is the **Heat Flux**) specified on the other wall. k_{w1} and k_{w2} are the thermal conductivities of the uncoupled thin walls. Note that the gap between the walls in Figure 6.44: Uncoupled Thin Walls (p. 376) is not part of the model; it is included in the figure only to show where the thermal boundary condition for each uncoupled wall is applied.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.44: Uncoupled Thin Walls



6.3.15.3.9. Shell Conduction

To enable shell conduction for a wall, enable the **Shell Conduction** option in the **Wall** boundary condition dialog box. You can then click the **Edit...** button to open the **Shell Conduction Layers** dialog box, where you can define the properties of the single or multiple layers of the shell. Note that you must specify a nonzero wall thickness for every layer of the shell. When shell conduction is enabled, ANSYS Fluent will compute heat conduction for the wall not only in the normal direction (which is always computed when the energy equation is solved), but also in the planar directions. The **Shell Conduction** option will appear in the **Wall** dialog box for all walls when solution of the energy equation is active (except for mapped interfaces). For information about how the thermal conditions are applied on a wall with shell conduction enabled, managing multiple shells, and postprocessing shell conduction walls, see Shell Conduction Considerations (p. 838).

ANSYS Fluent cases with shell conduction can be read in serial or parallel. Either a partitioned or an unpartitioned case file can be read in parallel (see Mesh Partitioning and Load Balancing (p. 2030) for more information on partitioning). After reading a case file in parallel, shell zones can be created on any wall.

To convert every wall with a finite thickness into a shell with a single action, the TUI command `define/boundary-conditions/modify-zones/create-all-shell-threads` can be used; each converted shell will have a single layer with the same thickness as the original thin wall (any pre-existing shells will not be modified). To disable shell conduction in every wall with a single action, the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TUI command `define/boundary-conditions/modify-zones/delete-all-shells` can be used. These capabilities are available in both serial and parallel mode.

Important

Note that the shell conduction model has several limitations:

- It cannot be applied on non-conformal interfaces, including mapped interfaces.
- It cannot be applied on moving wall zones.
- It cannot be used with FMG initialization.
- Shell conduction is not available when the wall is set up to receive thermal data via system coupling.
- It is available only in 3D.
- It is available only when the pressure-based solver is used.
- Shells cannot be split or merged. If you need to split or merge a shell, disable the **Shell Conduction** option for the wall, perform the split or merge operation, and then enable **Shell Conduction** for the new wall zones.
- The shell conduction model cannot be used on a wall zone that has been adapted. If you want to perform adaption elsewhere in the computational domain, be sure to use the mask register described in Manipulating Adaption Registers (p. 1701).
- Fluxes at the ends of a shell are not included in the heat balance reports. These fluxes are accounted for correctly in the ANSYS Fluent solution, but not in the flux report itself.
- The junction of a wall with shell conduction enabled and a non-conformal coupled wall is not supported. Such a junction will not be thermally connected, that is, there will be no heat transfer between the shell and the mesh interface wall.
- When running the parallel solver with the shell conduction model, note that coupled walls are encapsulated. If you encounter problems with the partitioning of the mesh, you can try changing the encapsulation method to see if that resolves the problem (see Troubleshooting (p. 2056) for this and other troubleshooting options).

6.3.15.3.10. Heat Transfer Boundary Conditions Through System Coupling

System coupling allows the input and output of thermal data from ANSYS Fluent. When Fluent is coupled with another system in Workbench using System Coupling, you can select the **via System Coupling** option on the desired wall boundaries to receive thermal data through System Coupling service. Note that this option does not need to be selected to provide thermal data from Fluent.

For more details about setting up a simulation with system coupling, see the Fluent in Workbench User's Guide and the System Coupling User's Guide.

When thermal data is transferred into Fluent via System Coupling, the following variables are available:

- temperature

- heat flow (heat rate)

When thermal data is transferred out of Fluent via System Coupling, the following variables are available:

- temperature
- heat flow (heat rate)
- heat transfer coefficient (also known as "convection coefficient")
- near wall temperature (also known as "bulk temperature" or "ambient temperature")

For each data transfer, you specify the type of data transfer (the variables transferred) during the System Coupling setup.

As part of standard heat transfer boundary condition settings in ANSYS Fluent, you can also specify the type of material, the thickness of the wall, and the heat generation rate in the wall. Select the material type in the **Material Name** drop-down list; if you want to check or modify the properties of the selected material, you can click **Edit...** to open the **Edit Material** dialog box (this dialog box contains just the properties of the selected material, and not the full contents of the standard **Create/Edit Materials** dialog box). You can specify the thickness in the **Wall Thickness** number-entry box and the heat generation rate in the **Heat Generation Rate** number-entry box.

Note

A boundary will behave in the same way as an adiabatic boundary if the **via System Coupling** option is enabled on this boundary, and ANSYS Fluent is not receiving data from the coupling data transfer. Fluent is not receiving coupling data if it is either not involved with a System Coupling simulation, or if the coupling is a one-way transfer with the Fluent analysis only providing data to the second solver.

6.3.15.3.11. Heat Transfer Boundary Conditions Across a Mapped Interface

Mapped interfaces provide a robust approach for modeling coupled walls between zones when the interface zones penetrate each other or have gaps between them (see Figure 6.45: 2D Interface with Penetration and Gaps (p. 379)). For the interface wall boundary zones created as part of such interfaces, **via Mapped Interface** is automatically selected from the **Thermal Conditions** list in the **Thermal** tab of the **Wall** dialog box, in order to interpolate the thermal data.

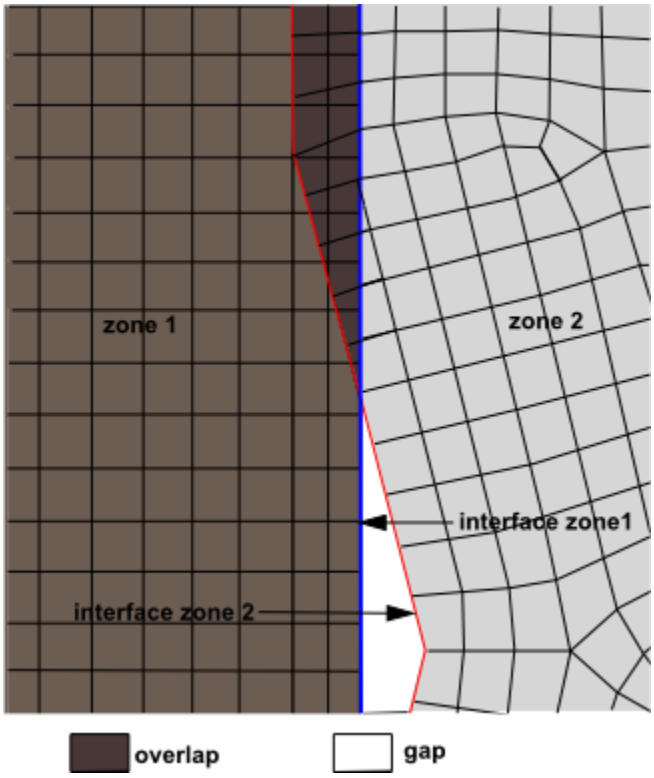
Important

Note that thermal coupling calculations will not be performed at mapped interfaces for the Eulerian multiphase model.

For more details about mapped interfaces, see The Mapped Option (p. 158) and Using a Non-Conformal Mesh in ANSYS Fluent (p. 164).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.45: 2D Interface with Penetration and Gaps



Such boundaries also allow you to specify the standard heat transfer boundary condition settings, such as the type of material, the thickness of the wall, and the heat generation rate in the wall. Select the material type in the **Material Name** drop-down list; if you want to check or modify the properties of the selected material, you can click **Edit...** to open the **Edit Material** dialog box (this dialog box contains just the properties of the selected material, and not the full contents of the standard **Create/Edit Materials** dialog box). You can specify the thickness in the **Wall Thickness** number-entry box and the heat generation rate in the **Heat Generation Rate** number-entry box. Note that the **Shell Conduction** option is not available with the **via Mapped Interface** thermal condition.

6.3.15.4. Species Boundary Conditions for Walls

By default, a zero-gradient condition for all species is assumed at walls (except for species that participate in surface reactions), but it is also possible to specify species mass fractions at walls. That is, Dirichlet boundary conditions such as those that are specified at inlets can be used at walls as well.

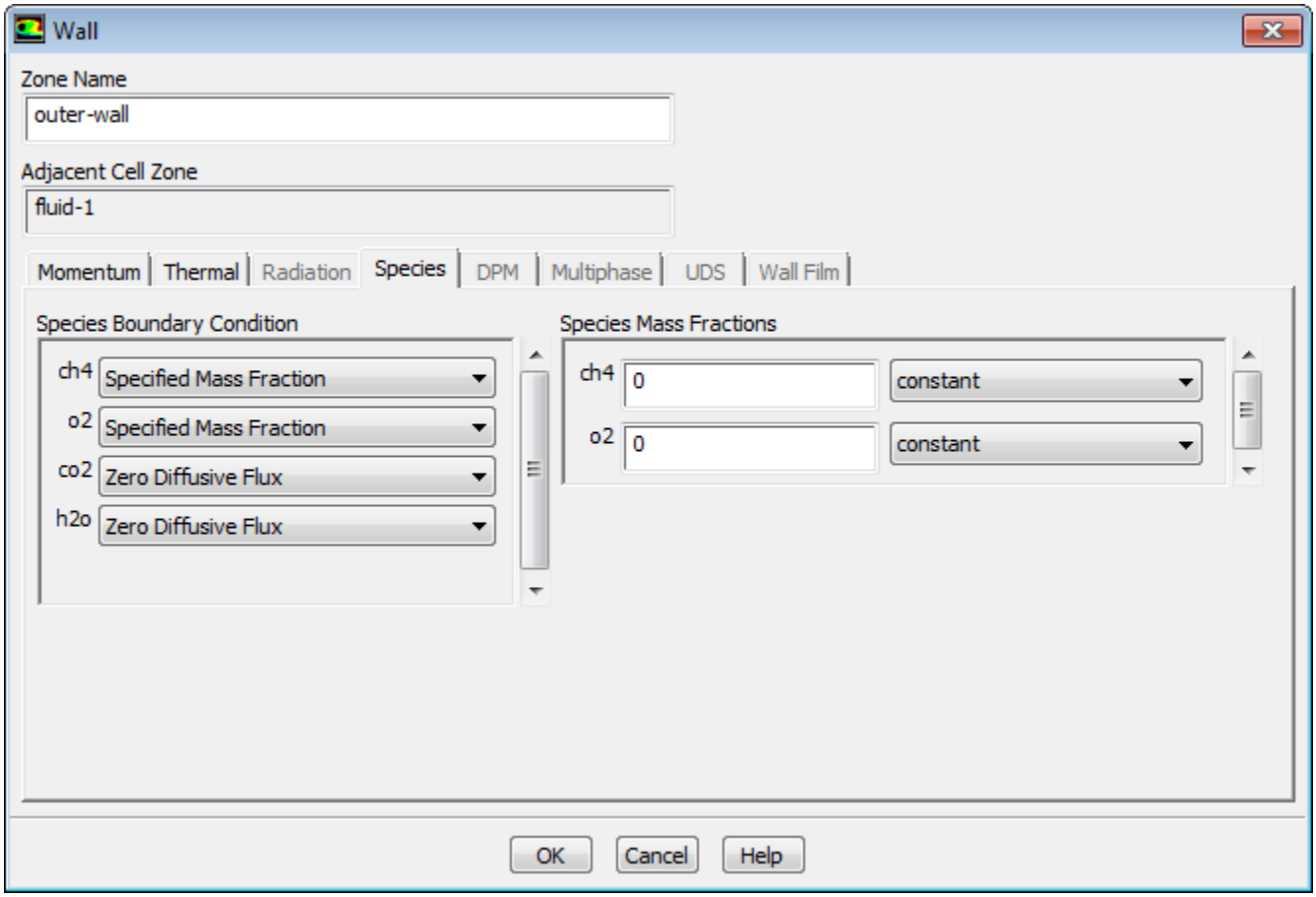
If you want to retain the default zero-gradient condition for a species, no inputs are required. If you want to specify the mass fraction for a species at the wall, the steps are as follows:

1. Click the **Species** tab in the Wall Dialog Box (p. 2456) to view the species boundary conditions for the wall (see Figure 6.46: The Wall Dialog Box for Species Boundary Condition Input (p. 380)).

2. Under **Species Boundary Condition**, select **Specified Mass Fraction** (rather than **Zero Diffusive Flux**) in the drop-down list to the right of the species name. The dialog box will expand to include a field for **Species Mass Fractions**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.46: The Wall Dialog Box for Species Boundary Condition Input



3. Under **Species Mass Fractions**, specify the mass fraction for the species.

The boundary condition type for each species is specified separately, so you can choose to use different methods for different species.

If you are modeling species transport with reactions, you can, alternatively, enable a reaction mechanism at a wall by turning on the **Reaction** option and selecting an available mechanism from the **Reaction Mechanisms** drop-down list. See Defining Zone-Based Reaction Mechanisms (p. 984) for more information about defining reaction mechanisms.

You can also model unresolved surface washcoats, which greatly increase the catalytic surface area, by specifying the **Surface Area Washcoat Factor**. The surface washcoat increases the area available for surface reaction.

6.3.15.4.1. Reaction Boundary Conditions for Walls

If you have enabled the modeling of wall surface reactions in the Species Model Dialog Box (p. 2216), you can indicate whether or not surface reactions should be activated for the wall. In the **Species** tab of the **Wall** dialog box (Figure 6.46: The Wall Dialog Box for Species Boundary Condition Input (p. 380)), turn the **Surface Reactions** option on or off.

Note that a zero-gradient condition is assumed at the wall for species that do not participate in any surface reactions.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.15.5. Radiation Boundary Conditions for Walls

If you are using the gray P-1, DTRM, DO, surface-to-surface, or MC models, you will need to set the emissivity of the wall (**Internal Emissivity**) in the **Thermal** tab of the **Wall** dialog box. If you are using the Rosseland model you do not need to set the emissivity, because ANSYS Fluent assumes the emissivity is 1.

For the non-gray P-1, DO, or MC models, specify a constant **Internal Emissivity** for each wavelength band in the **Radiation** tab of the **Wall** dialog box (the default value in each band is 1). Alternatively, you can specify the internal emissivity using a boundary condition parameter (see Creating a New Parameter (p. 246)). If you are using the non-gray DO model, you will also need to define the wall as opaque or semi-transparent in the **Radiation** tab. See Defining Boundary Conditions for Radiation (p. 869) for details. If you are using the MC models, you can enable a **Boundary Source** option, which allows you to specify the **Beam Direction** and the **Direct Irradiation**.

6.3.15.6. Discrete Phase Model (DPM) Boundary Conditions for Walls

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the wall in the **DPM** section of the **Wall** dialog box. See Setting Boundary Conditions for the Discrete Phase (p. 1301) for details.

6.3.15.6.1. Wall Adhesion Contact Angle for VOF Model

If you are using the VOF model and you are modeling wall adhesion, you can specify the contact angle for each pair of phases at the wall in the **Momentum** tab of the **Wall** dialog box. See Steps for Setting Boundary Conditions (p. 1391) for details.

6.3.15.7. User-Defined Scalar (UDS) Boundary Conditions for Walls

If you have defined UDS transport equations in your model, you can specify boundary conditions for each equation in the **UDS** section of the **Wall** dialog box. See Setting Up UDS Equations in ANSYS Fluent (p. 575) for details.

6.3.15.8. Wall Film Conditions for Walls

If you are using the Eulerian Wall Film model (see Modeling Eulerian Wall Films (p. 1535) for details), you can set liquid film conditions at the wall in the **Wall Film** tab of the **Wall** dialog box. This tab is available only if you have enabled the Eulerian Wall Film model in the Models Task Page (p. 2169).

For details, see Setting Eulerian Wall Film Solution Controls (p. 1539).

6.3.15.9. Default Settings at Wall Boundaries

The default thermal boundary condition is a fixed heat flux of zero. Walls are, by default, not moving.

6.3.15.10. Shear-Stress Calculation Procedure at Wall Boundaries

For no-slip wall conditions, ANSYS Fluent uses the properties of the flow adjacent to the wall/fluid boundary to predict the shear stress on the fluid at the wall. In laminar flows this calculation simply depends on the velocity gradient at the wall, while in turbulent flows one of the approaches described in Near-Wall Treatments for Wall-Bounded Turbulent Flows in the Theory Guide is used.

For specified-shear walls, ANSYS Fluent will compute the tangential velocity at the boundary.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you are modeling inviscid flow with ANSYS Fluent, all walls use a slip condition, so they are frictionless and exert no shear stress on the adjacent fluid.

6.3.15.10.1. Shear-Stress Calculation in Laminar Flow

In a laminar flow, the wall shear stress is defined by the normal velocity gradient at the wall as

$$\tau_w = \mu \frac{\partial v}{\partial n} \tag{6.117}$$

When there is a steep velocity gradient at the wall, you must be sure that the mesh is sufficiently fine to accurately resolve the boundary layer. Guidelines for the appropriate placement of the near-wall node in laminar flows are provided in Mesh Element Distribution (p. 133).

6.3.15.10.2. Shear-Stress Calculation in Turbulent Flows

Wall treatments for turbulent flows are described in Near-Wall Treatments for Wall-Bounded Turbulent Flows in the Theory Guide.

6.3.15.11. Heat Transfer Calculations at Wall Boundaries

6.3.15.11.1. Temperature Boundary Conditions

When a fixed temperature condition is applied at the wall, the heat flux to the wall from a fluid cell is computed as

$$q = h_f \left(T_w - T_f \right) + q_{rad} \tag{6.118}$$

where

h_f = fluid-side local heat transfer coefficient

T_w = wall surface temperature

T_f = local fluid temperature

q_{rad} = radiative heat flux

Note that the fluid-side heat transfer coefficient is computed based on the local flow-field conditions (for example, turbulence level, temperature, and velocity profiles), as described by Equation 6.125 (p. 384).

Heat transfer to the wall boundary from a solid cell is computed as

$$q = \frac{k_s}{\Delta n} \left(T_w - T_s \right) + q_{rad} \tag{6.119}$$

where

k_s = thermal conductivity of the solid

T_s = local solid temperature

Δn = distance between wall surface and the solid cell center

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.15.11.2. Heat Flux Boundary Conditions

When you define a heat flux boundary condition at a wall, you specify the heat flux at the wall surface. ANSYS Fluent uses Equation 6.118 (p. 382) and your input of heat flux to determine the wall surface temperature adjacent to a fluid cell as

$$T_w = \frac{q - q_{rad}}{h_f} + T_f \tag{6.120}$$

where, as noted above, the fluid-side heat transfer coefficient is computed based on the local flow-field conditions. When the wall borders a solid region, the wall surface temperature is computed as

$$T_w = \frac{\left(q - q_{rad}\right)\Delta n}{k_s} + T_s \tag{6.121}$$

6.3.15.11.3. Convective Heat Transfer Boundary Conditions

When you specify a convective heat transfer coefficient boundary condition at a wall, ANSYS Fluent uses your inputs of the external heat transfer coefficient and external heat sink temperature to compute the heat flux to the wall as

$$\begin{aligned} q &= h_f\left(T_w - T_f\right) + q_{rad} \\ &= h_{ext}\left(T_{ext} - T_w\right) \end{aligned} \tag{6.122}$$

where

h_{ext} = external heat transfer coefficient defined by you

T_{ext} = external heat-sink temperature defined by you

q_{rad} = radiative heat flux

Equation 6.122 (p. 383) assumes a wall of zero-thickness.

6.3.15.11.4. External Radiation Boundary Conditions

When the external radiation boundary condition is used in ANSYS Fluent, the heat flux to the wall is computed as

$$\begin{aligned} q &= h_f\left(T_w - T_f\right) + q_{rad} \\ &= \varepsilon_{ext}\sigma\left(T_\infty^4 - T_w^4\right) \end{aligned} \tag{6.123}$$

where

ε_{ext} = emissivity of the external wall surface defined by you

σ = Stefan-Boltzmann constant

T_w = surface temperature of the wall

T_∞ = temperature of the radiation source or sink on the exterior of the domain, defined by you

q_{rad} = radiative heat flux to the wall from within the domain

Equation 6.123 (p. 383) assumes a wall of zero-thickness.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.15.11.5. Combined External Convection and Radiation Boundary Conditions

When you choose the combined external heat transfer condition, the heat flux to the wall is computed as

$$\begin{aligned} q &= h_f\left(T_w - T_f\right) + q_{rad} \\ &= h_{ext}\left(T_{ext} - T_w\right) + \varepsilon_{ext}\sigma\left(T_\infty^4 - T_w^4\right) \end{aligned} \tag{6.124}$$

where the variables are as defined above. Equation 6.124 (p. 384) assumes a wall of zero-thickness.

6.3.15.11.6. Calculation of the Fluid-Side Heat Transfer Coefficient

In laminar flows, the fluid side heat transfer at walls is computed using Fourier's law applied at the walls. ANSYS Fluent uses its discrete form:

$$q = k_f\left(\frac{\partial T}{\partial n}\right)_{wall} \tag{6.125}$$

where n is the local coordinate normal to the wall.

For turbulent flows, ANSYS Fluent uses the law-of-the-wall for temperature derived using the analogy between heat and momentum transfer [56] (p. 2890). See Standard Wall Functions in the separate Theory Guide for details.

6.3.16. Symmetry Boundary Conditions

Symmetry boundary conditions are used when the physical geometry of interest, and the expected pattern of the flow/thermal solution, have mirror symmetry. They can also be used to model zero-shear slip walls in viscous flows. This section describes the treatment of the flow at symmetry planes and provides examples of the use of symmetry. You do not define any boundary conditions at symmetry boundaries, but you must take care to correctly define your symmetry boundary locations.

Important

At the centerline of an axisymmetric geometry, you should use the axis boundary type rather than the symmetry boundary type, as illustrated in Figure 6.54: Use of an Axis Boundary as the Centerline in an Axisymmetric Geometry (p. 389). See Axis Boundary Conditions (p. 389) for details.

6.3.16.1. Examples of Symmetry Boundaries

Symmetry boundaries are used to reduce the extent of your computational model to a symmetric subsection of the overall physical system. Figure 6.47: Use of Symmetry to Model One Quarter of a 3D Duct (p. 385) and Figure 6.48: Use of Symmetry to Model One Quarter of a Circular Cross-Section (p. 385) illustrate two examples of symmetry boundary conditions used in this way.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.47: Use of Symmetry to Model One Quarter of a 3D Duct

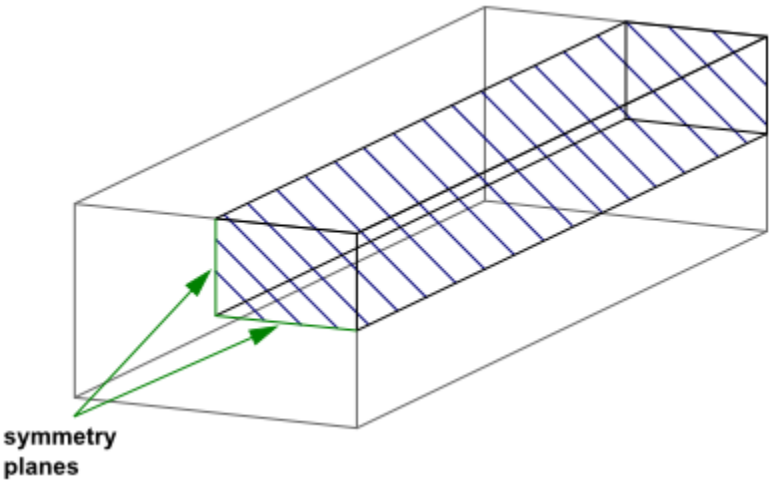


Figure 6.48: Use of Symmetry to Model One Quarter of a Circular Cross-Section

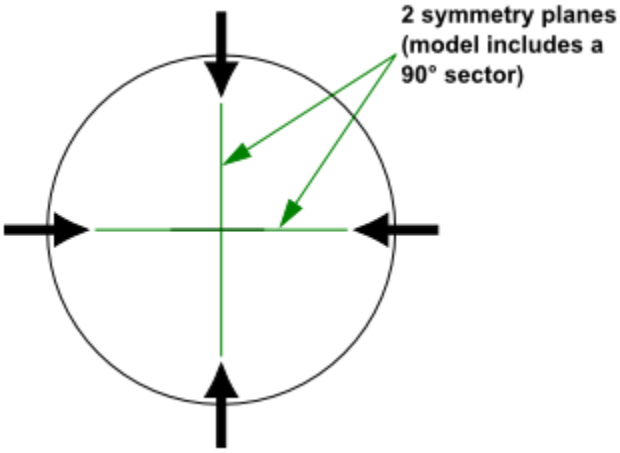
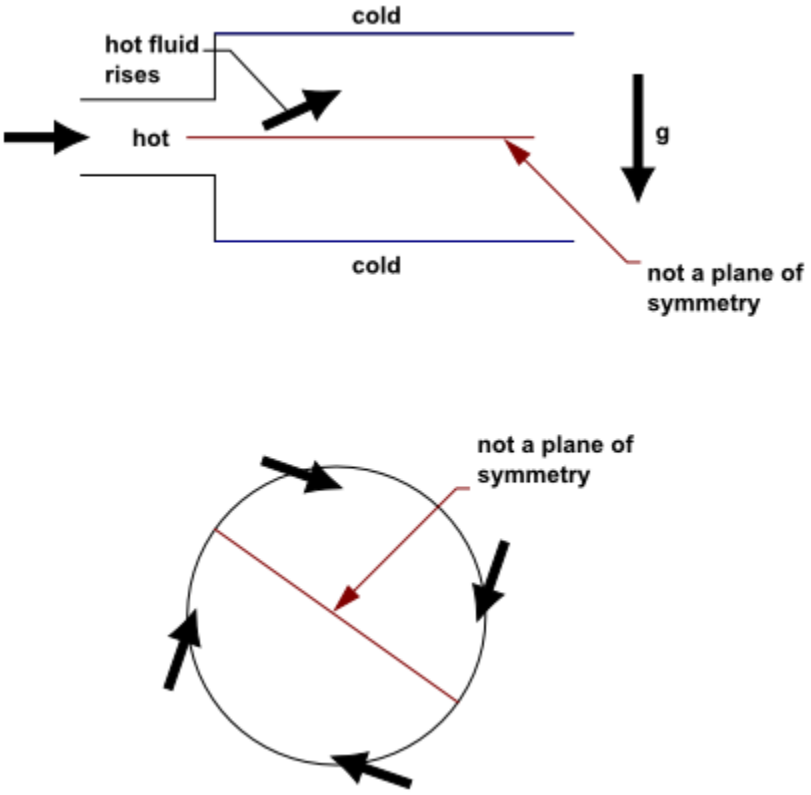


Figure 6.49: Inappropriate Use of Symmetry (p. 386) illustrates two problems in which a symmetry plane would be *inappropriate*. In both examples, the problem geometry is symmetric but the flow itself does not obey the symmetry boundary conditions. In the first example, buoyancy creates an asymmetric flow. In the second, swirl in the flow creates a flow normal to the would-be symmetry plane. Note that this second example should be handled using rotationally periodic boundaries (as illustrated in Figure 6.50: Use of Periodic Boundaries to Define Swirling Flow in a Cylindrical Vessel (p. 387)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.49: Inappropriate Use of Symmetry



6.3.16.2. Calculation Procedure at Symmetry Boundaries

ANSYS Fluent assumes a zero flux of all quantities across a symmetry boundary. There is no convective flux across a symmetry plane: the normal velocity component at the symmetry plane is therefore zero. There is no diffusion flux across a symmetry plane: the normal gradients of all flow variables are therefore zero at the symmetry plane. The symmetry boundary condition can therefore be summarized as follows:

- zero normal velocity at a symmetry plane
- zero normal gradients of all variables at a symmetry plane

As stated above, these conditions determine a zero flux across the symmetry plane, which is required by the definition of symmetry. Since the shear stress is zero at a symmetry boundary, it can also be interpreted as a "slip" wall when used in viscous flow calculations.

6.3.17. Periodic Boundary Conditions

Periodic boundary conditions are used when the physical geometry of interest and the expected pattern of the flow/thermal solution have a periodically repeating nature. Two types of periodic conditions are available in ANSYS Fluent. The first type does not allow a pressure drop across the periodic planes. (Note to FLUENT 4 users: This type of periodic boundary is referred to as a "cyclic" boundary in FLUENT 4.) The second type allows a pressure drop to occur across translationally periodic boundaries, enabling you to model "fully-developed" periodic flow. (In FLUENT 4 this is a "periodic" boundary.)

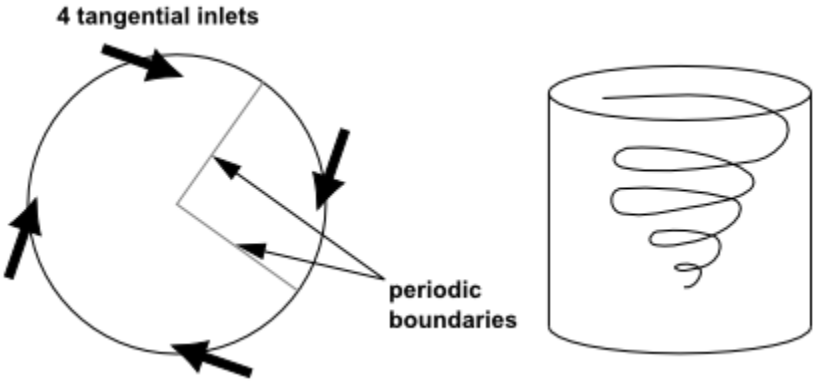
This section discusses the no-pressure-drop periodic boundary condition. A complete description of the fully-developed periodic flow modeling capability is provided in Periodic Flows (p. 582).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.17.1. Examples of Periodic Boundaries

Periodic boundary conditions are used when the flows across two opposite planes in your computational model are identical. Figure 6.50: Use of Periodic Boundaries to Define Swirling Flow in a Cylindrical Vessel (p. 387) illustrates a typical application of periodic boundary conditions. In this example the flow entering the computational model through one periodic plane is identical to the flow exiting the domain through the opposite periodic plane. Periodic planes are always used in pairs as illustrated in this example.

Figure 6.50: Use of Periodic Boundaries to Define Swirling Flow in a Cylindrical Vessel



6.3.17.2. Inputs for Periodic Boundaries

For a periodic boundary without any pressure drop, there is only one input you need to consider: whether the geometry is rotationally or translationally periodic. (Additional inputs are required for a periodic flow with a periodic pressure drop. See Periodic Flows (p. 582).)

Rotationally periodic boundaries are boundaries that form an included angle about the centerline of a rotationally symmetric geometry. Figure 6.50: Use of Periodic Boundaries to Define Swirling Flow in a Cylindrical Vessel (p. 387) illustrates rotational periodicity. Translationally periodic boundaries are boundaries that form periodic planes in a rectilinear geometry. Figure 6.51: Example of Translational Periodicity - Physical Domain (p. 387) illustrates translationally periodic boundaries.

Figure 6.51: Example of Translational Periodicity - Physical Domain

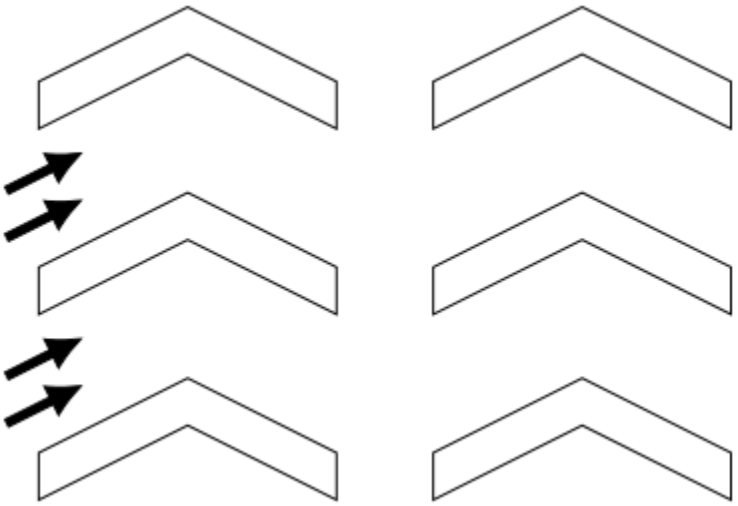
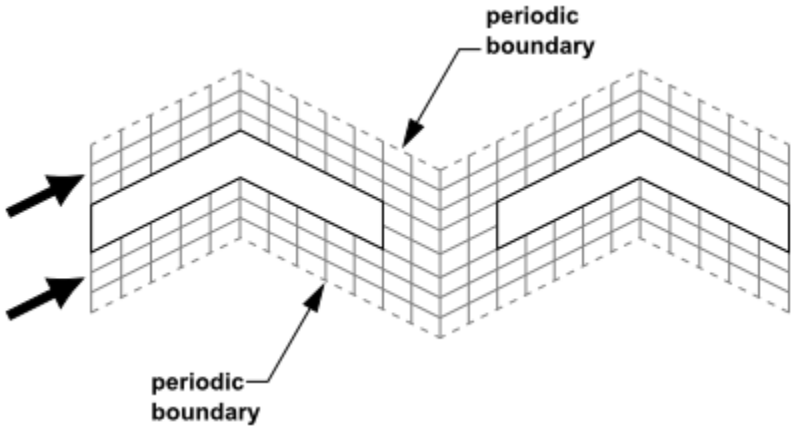
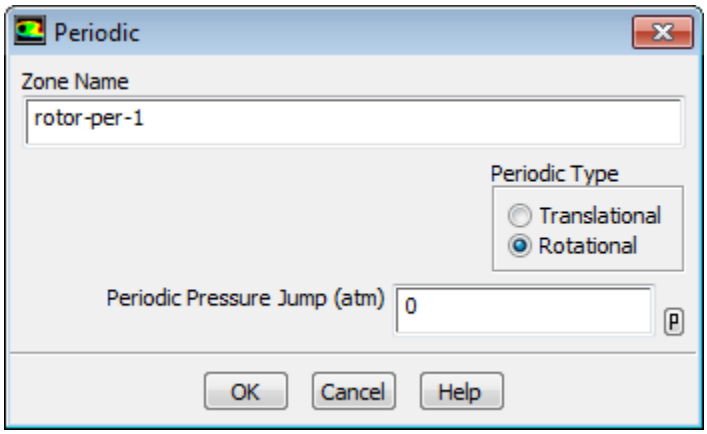


Figure 6.52: Example of Translational Periodicity - Modeled Domain



You will specify translational or rotational periodicity for a periodic boundary in the Periodic Dialog Box (p. 2427) (Figure 6.53: The Periodic Dialog Box (p. 388)), which is opened from the **Boundary Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 241)).

Figure 6.53: The Periodic Dialog Box



Note that there will be an additional item in the **Periodic** dialog box for the density-based solver, which allows you to specify the periodic pressure jump. See Periodic Flows (p. 582) for details.

If the domain is rotationally periodic, select **Rotational** as the **Periodic Type**; if it is translationally periodic, select **Translational**. For rotationally periodic domains, the solver will automatically compute the angle through which the periodic zone is rotated. The axis used for this rotation is the axis of rotation specified for the adjacent cell zone.

Note that there is no need for the adjacent cell zone to be moving for you to use a rotationally periodic boundary. You could, for example, model pipe flow in 3D using a stationary reference frame with a pie-slice of the pipe; the sides of the slice would require rotational periodicity.

Click the **Check** button in the **Mesh** group box of the **Setting Up Domain** tab (see Checking the Mesh (p. 192)) to compute and display the minimum, maximum, and average rotational angles of all faces on periodic boundaries. If the difference between the minimum, maximum, and average values is not negligible, then there is a problem with the mesh: the mesh geometry is not periodic about the specified axis.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.17.3. Default Settings at Periodic Boundaries

By default, all periodic boundaries are translational.

6.3.17.4. Calculation Procedure at Periodic Boundaries

ANSYS Fluent treats the flow at a periodic boundary as though the opposing periodic plane is a direct neighbor to the cells adjacent to the first periodic boundary. Therefore, when calculating the flow through the periodic boundary adjacent to a fluid cell, the flow conditions at the fluid cell adjacent to the opposite periodic plane are used.

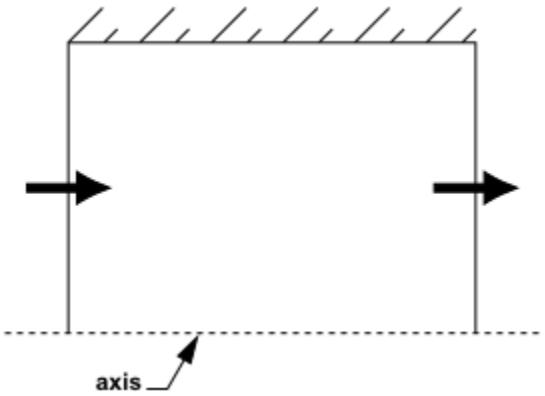
6.3.18. Axis Boundary Conditions

The axis boundary type must be used as the centerline of an axisymmetric geometry (see Figure 6.54: Use of an Axis Boundary as the Centerline in an Axisymmetric Geometry (p. 389)). It can also be used for the centerline of a cylindrical-polar quadrilateral or hexahedral mesh (for example, a mesh created for a structured-mesh code such as FLUENT 4). You do not need to define any boundary conditions at axis boundaries.

Important

When creating 2D axisymmetric geometry, the axis boundary must lie on the y=0 line.

Figure 6.54: Use of an Axis Boundary as the Centerline in an Axisymmetric Geometry



6.3.18.1. Calculation Procedure at Axis Boundaries

To determine the appropriate physical value for a particular variable at a point on the axis, ANSYS Fluent uses the cell value in the adjacent cell.

6.3.19. Fan Boundary Conditions

The fan model is a lumped parameter model that can be used to determine the impact of a fan with known characteristics upon some larger flow field. The fan boundary type allows you to input an empirical fan curve that governs the relationship between head (pressure rise) and flow rate (velocity) across a fan element. You can also specify radial and tangential components of the fan swirl velocity. The fan model does not provide an accurate description of the detailed flow through the fan blades. Instead, it predicts the amount of flow through the fan. Fans may be used in conjunction with other

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

flow sources, or as the sole source of flow in a simulation. In the latter case, the system flow rate is determined by the balance between losses in the system and the fan curve.

ANSYS Fluent also provides a connection for a special user-defined fan model that updates the pressure jump function during the calculation. This feature is described in User-Defined Fan Model (p. 427).

6.3.19.1. Fan Equations

6.3.19.1.1. Modeling the Pressure Rise Across the Fan

A fan is considered to be infinitely thin, and the discontinuous pressure rise across it is specified as a function of the velocity through the fan. The relationship may be a constant, a polynomial, piecewise-linear, or piecewise-polynomial function, or a user-defined function.

In the case of a polynomial, the relationship is of the form

$$\Delta p = \sum_{n=1}^{N} f_n v^{n-1} \tag{6.126}$$

where Δp is the pressure jump, f_n are the pressure-jump polynomial coefficients, and v is the magnitude of the local fluid velocity normal to the fan.

Important

The velocity v can be either positive or negative. You must be careful to model the fan so that a pressure rise occurs for forward flow through the fan.

You can, optionally, use the mass-averaged velocity normal to the fan to determine a single pressure-jump value for all faces in the fan zone.

6.3.19.1.2. Modeling the Fan Swirl Velocity

For three-dimensional problems, the values of the convected tangential and radial velocity fields can be imposed on the fan surface to generate swirl. These velocities can be specified as functions of the radial distance from the fan center. The relationships may be constant or polynomial functions, or user-defined functions.

Important

You must use SI units for all fan swirl velocity inputs.

For the case of polynomial functions, the tangential and radial velocity components can be specified by the following equations:

$$U_\theta = \sum_{n=-1}^{N} f_n r^n; -1 \le N \le 6 \tag{6.127}$$

$$U_r = \sum_{n=-1}^{N} g_n r^n; -1 \le N \le 6 \tag{6.128}$$

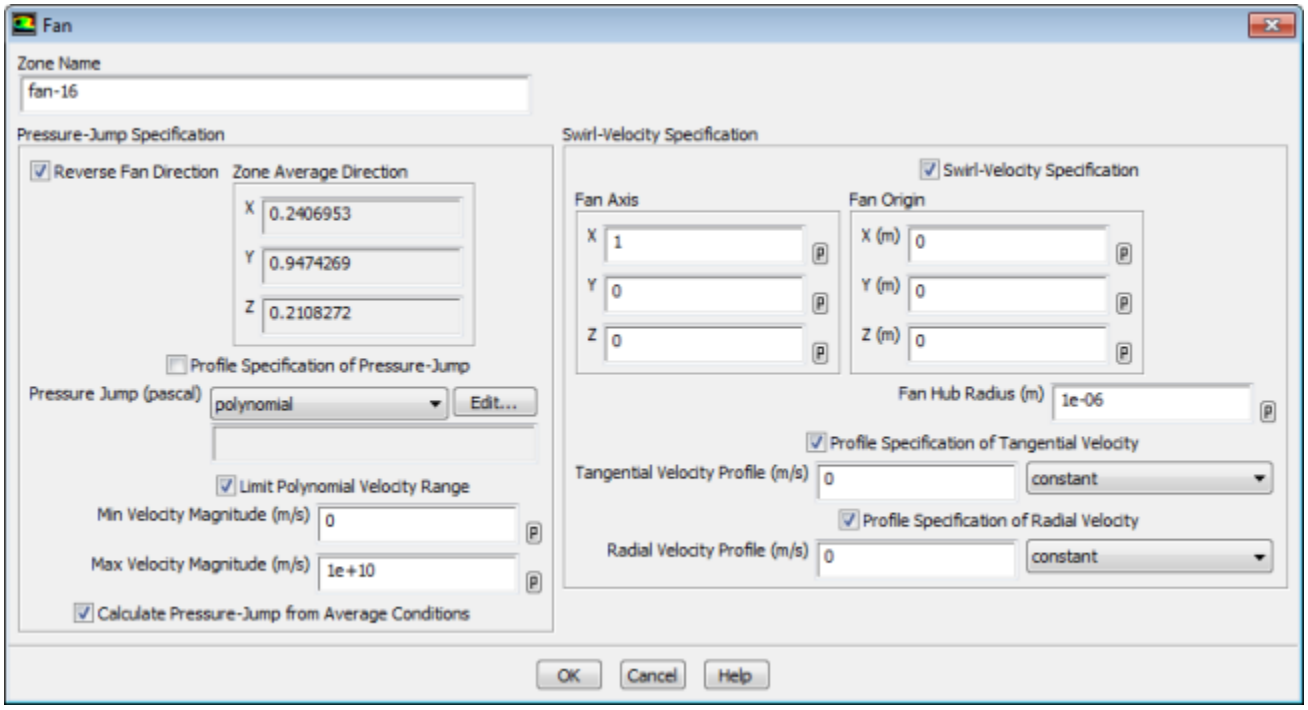
where U_θ and U_r are, respectively, the tangential and radial velocities on the fan surface in m/s, f_n and g_n are the tangential and radial velocity polynomial coefficients, and r is the distance to the fan center.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.19.2. User Inputs for Fans

Once the fan zone has been identified (in the **Boundary Conditions** task page), you will set all modeling inputs for the fan in the Fan Dialog Box (p. 2397) (Figure 6.55: The Fan Dialog Box (p. 391)), which is opened from the Boundary Conditions Task Page (p. 2389) (as described in Setting Cell Zone and Boundary Conditions (p. 241)).

Figure 6.55: The Fan Dialog Box



Inputs for a fan are as follows:

1. Identify the fan zone.
2. Define the pressure jump across the fan.
3. Define the discrete phase boundary condition for the fan (for discrete phase calculations).
4. Define the swirl velocity, if desired (3D only).

6.3.19.2.1. Identifying the Fan Zone

Since the fan is considered to be infinitely thin, it must be modeled as the interface between cells, rather than a cell zone. Therefore the fan zone is a type of internal face zone (where the faces are line segments in 2D or triangles/quadrilaterals in 3D). If, when you read your mesh into ANSYS Fluent, the fan zone is identified as an **interior** zone, right-click the zone in the tree and use the **Type** sub-menu to change it to a **fan** zone (for more details, see Changing Cell and Boundary Zone Types (p. 239)).

Once the interior zone has been changed to a fan zone, you can open the **Fan** dialog box and specify the pressure jump and, optionally, the swirl velocity.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.19.2.2. Defining the Pressure Jump

To define the pressure jump, you will specify a polynomial, piecewise-linear, or piecewise-polynomial function of velocity, a user-defined function, or a constant value. You should also check the **Zone Average Direction** vector to be sure that a pressure rise occurs for forward flow through the fan. The **Zone Average Direction**, calculated by the solver, is the face-averaged direction vector for the fan zone. If this vector is pointing in the direction you want the fan to blow, do not select **Reverse Fan Direction**; if it is pointing in the opposite direction, select **Reverse Fan Direction**.

6.3.19.2.2.1. Polynomial, Piecewise-Linear, or Piecewise-Polynomial Function

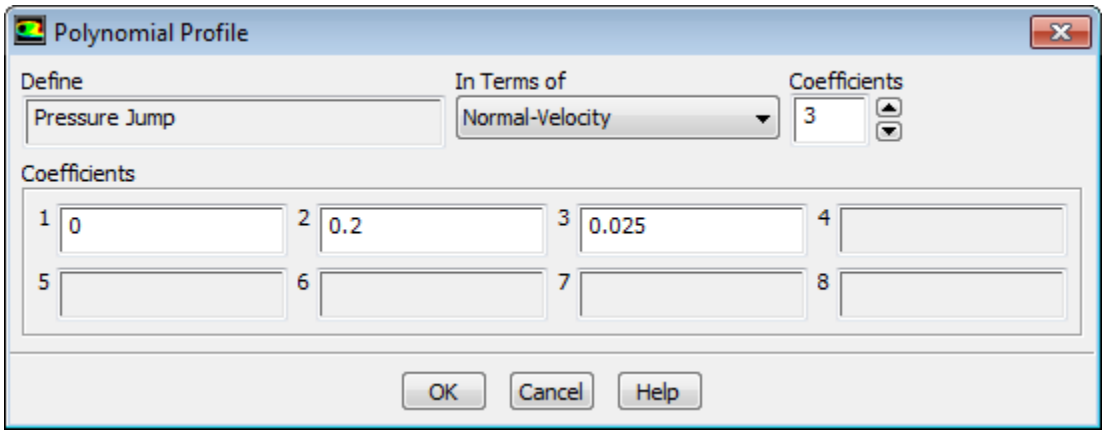
Follow these steps to set a polynomial, piecewise-linear, or piecewise-polynomial function for the pressure jump:

1. Check that the **Profile Specification of Pressure-Jump** option is off in the Fan Dialog Box (p. 2397).

2. Choose **polynomial**, **piecewise-linear**, or **piecewise-polynomial** in the drop-down list to the right of **Pressure-Jump**. (If the function type you want is already selected, you can click the **Edit...** button to open the dialog box where you will define the function.)

3. In the dialog box that appears for the definition of the **Pressure Jump** function (for example, Figure 6.56: Polynomial Profile Dialog Box for Pressure Jump Definition (p. 392)), enter the appropriate values. These profile input dialog boxes are used the same way as the profile input dialog boxes for temperature-dependent properties. See Defining Properties Using Temperature-Dependent Functions (p. 475) to find out how to use them.

Figure 6.56: Polynomial Profile Dialog Box for Pressure Jump Definition



4. Set any of the optional parameters described below. (optional)

When you define the pressure jump using any of these types of functions, you can choose to limit the minimum and maximum velocity magnitudes used to calculate the pressure jump. Enabling the **Limit Polynomial Velocity Range** option limits the pressure jump when a **Min Velocity Magnitude** and a **Max Velocity Magnitude** are specified.

Important

The values corresponding to the **Min Velocity Magnitude** and the **Max Velocity Magnitude** do not limit the flow field velocity to this range. However, this range does limit the value of

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the pressure jump, which is a polynomial and a function of velocity, as seen in Equation 6.126 (p. 390). If the calculated normal velocity magnitude exceeds the **Max Velocity Magnitude** that has been specified, then the pressure jump at the **Max Velocity Magnitude** value will be used. Similarly, if the calculated velocity is less than the specified **Min Velocity Magnitude**, the pressure jump at the **Min Velocity Magnitude** will be substituted for the pressure jump corresponding to the calculated velocity.

You also have the option to use the mass-averaged velocity normal to the fan to determine a single pressure-jump value for all faces in the fan zone. Turning on **Calculate Pressure-Jump from Average Conditions** enables this option.

6.3.19.2.2.2. Constant Value

To define a constant pressure jump, follow these steps:

1. Turn off the **Profile Specification of Pressure-Jump** option in the Fan Dialog Box (p. 2397).
2. Choose **constant** in the drop-down list to the right of **Pressure-Jump**.
3. Enter the value for Δp in the **Pressure-Jump** field.

You can follow the procedure below, if it is more convenient:

1. Turn on the **Profile Specification of Pressure-Jump** option.
2. Select **constant** in the drop-down list below **Pressure Jump Profile**, and enter the value for Δp in the **Pressure Jump Profile** field.

6.3.19.2.2.3. User-Defined Function or Profile

For a user-defined pressure-jump function or a function defined in a boundary profile file, you will follow these steps:

1. Turn on the **Profile Specification of Pressure-Jump** option.
2. Choose the appropriate function in the drop-down list below **Pressure Jump Profile**.

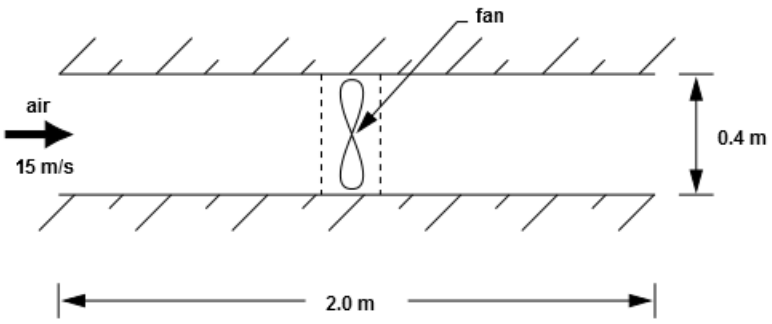
See the Fluent Customization Manual for information about user-defined functions, and Profiles (p. 435) for details about profile files.

6.3.19.2.2.4. Example: Determining the Pressure Jump Function

This example shows you how to determine the function for the pressure jump. Consider the simple two-dimensional duct flow illustrated in Figure 6.57: A Fan Located In a 2D Duct (p. 394). Air at constant density enters the 2.0 m × 0.4 m duct with a velocity of 15 m/s. Centered in the duct is a fan.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.57: A Fan Located In a 2D Duct



Assume that the fan characteristics are as follows when the fan is operating at 2000 rpm:

Q (m^3/s)	Δp (Pa)
25	0.0
20	175
15	350
10	525
5	700
0	875

where Q is the flow through the fan and Δp is the pressure rise across the fan. The fan characteristics in this example follow a simple linear relationship between pressure rise and flow rate. To convert this into a relationship between pressure rise and velocity, the cross-sectional area of the fan must be known. In this example, assuming that the duct is 1.0 m deep, this area is 0.4 m^2, so that the corresponding velocity values are as follows:

v (m/s)	Δp (Pa)
62.5	0.0
50.0	175
37.5	350
25.0	525
12.5	700
0	875

The polynomial form of this relationship is the following equation for a line:

$$\Delta p = 875 - 14v \tag{6.129}$$

6.3.19.2.3. Defining Discrete Phase Boundary Conditions for the Fan

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the fan. See Setting Boundary Conditions for the Discrete Phase (p. 1301) for details.

6.3.19.2.4. Defining the Fan Swirl Velocity

If you want to set tangential and radial velocity fields on the fan surface to generate swirl in a 3D problem, follow these steps:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Turn on the **Swirl-Velocity Specification** option in the Fan Dialog Box (p. 2397).
2. Specify the fan's axis of rotation by defining the axis origin (**Fan Origin**) and direction vector (**Fan Axis**).
3. Set the value for the radius of the fan's hub (**Fan Hub Radius**). The default is 1×10^{-6} to avoid division by zero in the polynomial.
4. Set the tangential and radial velocity functions as polynomial functions of radial distance, constant values, or user-defined functions.

Important

You must use SI units for all fan swirl velocity inputs.

6.3.19.2.4.1. Polynomial Function

To define a polynomial function for tangential or radial velocity, follow the steps below:

1. Check that the **Profile Specification of Tangential Velocity** or **Profile Specification of Radial Velocity** option is off in the Fan Dialog Box (p. 2397).
2. Enter the coefficients f_n in Equation 6.127 (p. 390) or g_n in Equation 6.128 (p. 390) in the **Tangential-** or **Radial-Velocity Polynomial Coefficients** field. Enter f_{-1} first, then f_0, and so on. Separate each coefficient by a blank space. Remember that the first coefficient is for $\frac{1}{r}$.

6.3.19.2.4.2. Constant Value

To define a constant tangential or radial velocity, the steps are as follows:

1. Turn on the **Profile Specification of Tangential Velocity** or **Profile Specification of Radial Velocity** option in the Fan Dialog Box (p. 2397).
2. Select **constant** in the drop-down list under **Tangential** or **Radial Velocity Profile**.
3. Enter the value for U_θ or U_r in the **Tangential** or **Radial Velocity Profile** field.

You can follow the procedure below, if it is more convenient:

1. Turn off the **Profile Specification of Tangential Velocity** or **Profile Specification of Radial Velocity** option in the Fan Dialog Box (p. 2397).
2. Enter the value for U_θ or U_r in the **Tangential-** or **Radial-Velocity Polynomial Coefficients** field.

6.3.19.2.4.3. User-Defined Function or Profile

For a user-defined tangential or radial velocity function or a function contained in a profile file, follow the procedure below:

1. Turn on the **Profile Specification of Tangential** or **Radial Velocity** option.
2. Choose the appropriate function from the drop-down list under **Tangential** or **Radial Velocity Profile**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See the Fluent Customization Manual for information about user-defined functions, and Profiles (p. 435) for details about profile files.

6.3.19.3. Postprocessing for Fans

6.3.19.3.1. Reporting the Pressure Rise Through the Fan

You can use the Surface Integrals Dialog Box (p. 2616) to report the pressure rise through the fan, as described in Surface Integration (p. 1920). There are two steps to this procedure:

1. Create a surface on each side of the fan zone. Use the Transform Surface Dialog Box (p. 2812) (as described in Transforming Surfaces (p. 1732)) to translate the fan zone slightly upstream and slightly downstream to create two new surfaces.

2. In the **Surface Integrals** dialog box, report the average **Static Pressure** just upstream and just downstream of the fan. You can then calculate the pressure rise through the fan.

6.3.19.3.2. Graphical Plots

Graphical reports of interest with fans are as follows:

- Contours or profiles of **Static Pressure** and **Static Temperature**.

- XY plots of **Static Pressure** and **Static Temperature** vs position.

Displaying Graphics (p. 1747) explains how to generate graphical displays of data.

Important

When generating these plots, be sure to turn off the display of node values so that you can see the different values on each side of the fan. (If you display node values, the cell values on either side of the fan will be averaged to obtain a node value, and you will not see, for example, the pressure jump across the fan.)

6.3.20. Radiator Boundary Conditions

A lumped-parameter model for a heat exchange element (for example, a radiator or condenser), is available in ANSYS Fluent. The radiator boundary type allows you to specify both the pressure drop and heat transfer coefficient as functions of the velocity normal to the radiator.

A more detailed heat exchanger model is also available in ANSYS Fluent. See Modeling Heat Exchangers (p. 919) for details.

6.3.20.1. Radiator Equations

6.3.20.1.1. Modeling the Pressure Loss Through a Radiator

A radiator is considered to be infinitely thin, and the pressure drop through the radiator is assumed to be proportional to the dynamic head of the fluid, with an empirically determined loss coefficient that you supply. That is, the pressure drop, Δp, varies with the normal component of velocity through the radiator, v, as follows:

$$\Delta p = k_L \frac{1}{2} \rho v^2 \tag{6.130}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where ρ is the fluid density, and k_L is the non-dimensional loss coefficient, which can be specified as a constant or as a polynomial, piecewise-linear, or piecewise-polynomial function.

In the case of a polynomial, the relationship is of the form

$$k_L = \sum_{n=1}^{N} r_n v^{n-1} \tag{6.131}$$

where r_n are polynomial coefficients and v is the magnitude of the local fluid velocity normal to the radiator.

6.3.20.1.2. Modeling the Heat Transfer Through a Radiator

The heat flux from the radiator to the surrounding fluid is given as

$$q = h\left(T_{ext} - T_{air,d}\right) \tag{6.132}$$

where q is the heat flux, $T_{air,d}$ is the temperature downstream of the heat exchanger (radiator), and T_{ext} is the radiator temperature. The convective heat transfer coefficient, h, can be specified as a constant or as a polynomial, piecewise-linear, or piecewise-polynomial function.

For a polynomial, the relationship is of the form

$$h = \sum_{n=0}^{N} h_n v^n \quad ; \quad 0 \le N \le 7 \tag{6.133}$$

where h_n are polynomial coefficients and v is the magnitude of the local fluid velocity normal to the radiator in m/s.

To define the actual heat flux $\left(q\right)$, specify a **Temperature** of absolute zero (0 K, 0 °R, –273.15 °C, –459.67 °F), and set the constant **Heat Flux** value.

To define the radiator temperature, enter the value for T_{ext} in the **Temperature** field. To define the **Heat-Transfer-Coefficient** $\left(h\right)$, you can specify a polynomial, piecewise-linear, or piecewise-polynomial function of velocity, or a constant value.

q (either the entered value or the value calculated using Equation 6.132 (p. 397)) is integrated over the radiator surface area.

Fluent does not allow you to specify a value less than absolute zero for the radiator temperature.

6.3.20.1.2.1. Calculating the Heat Transfer Coefficient

To model the thermal behavior of the radiator, you must supply an expression for the heat transfer coefficient, h, as a function of the fluid velocity through the radiator, v. To obtain this expression, consider the heat balance equation:

$$q = \frac{\dot{m} c_p \Delta T}{A} = h\left(T_{air,d} - T_{ext}\right) \tag{6.134}$$

where

q = heat flux (W/m^2)

$\dot{m}$ = fluid mass flow rate (kg/s)

c_p = specific heat capacity of fluid (J/kg-K)

h = empirical heat transfer coefficient (W/m^2-K)

T_{ext} = external temperature (reference temperature for the liquid) (K)

$T_{air,d}$ = temperature downstream from the heat exchanger (K)

A = heat exchanger frontal area (m^2)

Equation 6.134 (p. 397) can be rewritten as

$$q=\frac{\dot{m}c_p\left(T_{air,u}-T_{air,d}\right)}{A}=h\left(T_{air,d}-T_{ext}\right) \tag{6.135}$$

where $T_{air,u}$ is the upstream air temperature. The heat transfer coefficient, h, can therefore be computed as

$$h=\frac{\dot{m}c_p\left(T_{air,u}-T_{air,d}\right)}{A\left(T_{air,d}-T_{ext}\right)} \tag{6.136}$$

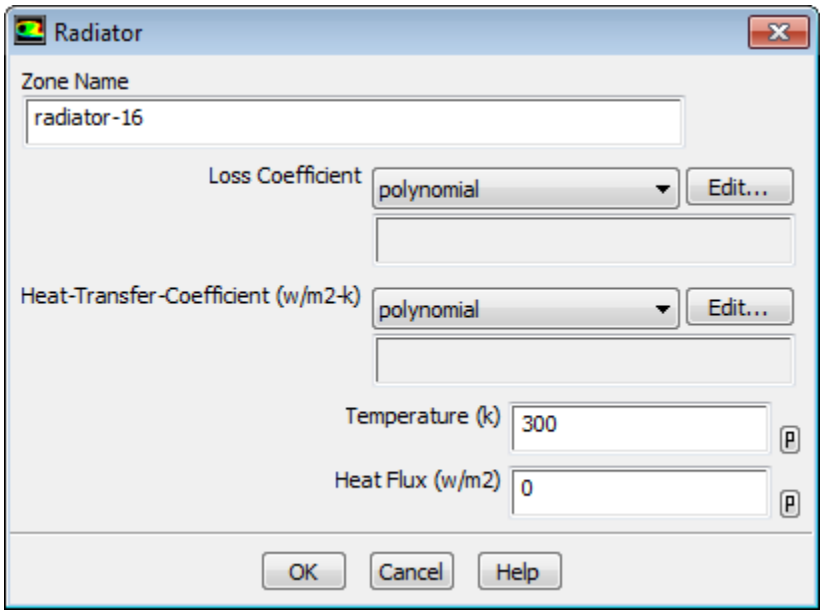
or, in terms of the fluid velocity,

$$h=\frac{\rho v c_p\left(T_{air,u}-T_{air,d}\right)}{T_{air,d}-T_{ext}} \tag{6.137}$$

6.3.20.2. User Inputs for Radiators

Once the radiator zone has been identified (in the **Boundary Conditions** task page), you will set all modeling inputs for the radiator in the Radiator Dialog Box (p. 2445) (Figure 6.58: The Radiator Dialog Box (p. 398)), which is opened from the Boundary Conditions Task Page (p. 2389) (as described in Setting Cell Zone and Boundary Conditions (p. 241)).

Figure 6.58: The Radiator Dialog Box



The inputs for a radiator are as follows:

1. Identify the radiator zone.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Define the pressure loss coefficient.

3. Define either the heat flux or the heat transfer coefficient and radiator temperature.

   If defining the heat flux, specify the **Temperature** *as absolute zero.*

4. Define the discrete phase boundary condition for the radiator (for discrete phase calculations).

6.3.20.2.1. Identifying the Radiator Zone

Since the radiator is considered to be infinitely thin, it must be modeled as the interface between cells, rather than a cell zone. Therefore the radiator zone is a type of internal face zone (where the faces are line segments in 2D or triangles/quadrilaterals in 3D). If, when you read your mesh into ANSYS Fluent, the radiator zone is identified as an **interior** zone, right-click the zone in the tree and use the **Type** sub-menu to change it to a **radiator** zone (for more details, see Changing Cell and Boundary Zone Types (p. 239)).

Once the interior zone has been changed to a radiator zone, you can open the **Radiator** dialog box and specify the loss coefficient and heat flux information.

6.3.20.2.2. Defining the Pressure Loss Coefficient Function

To define the pressure loss coefficient k_L you can specify a polynomial, piecewise-linear, or piecewise-polynomial function of velocity, or a constant value.

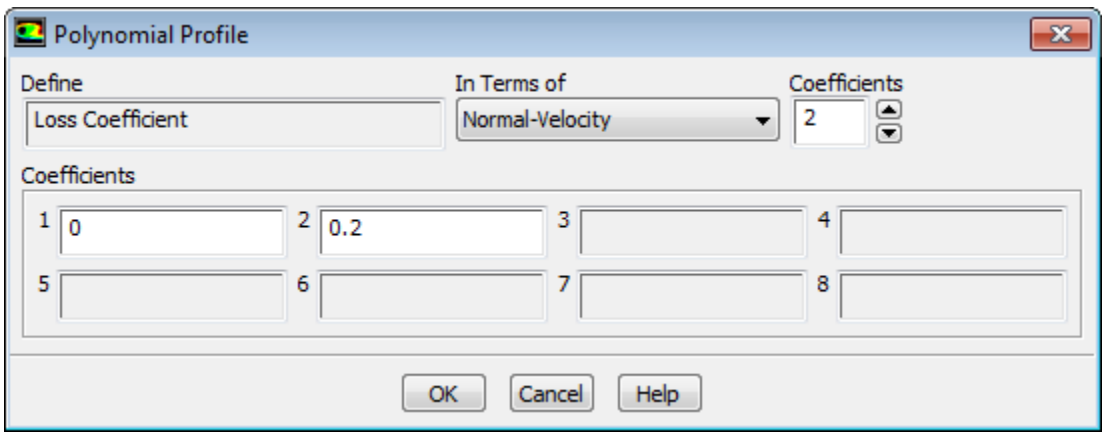
6.3.20.2.2.1. Polynomial, Piecewise-Linear, or Piecewise-Polynomial Function

Follow these steps to set a polynomial, piecewise-linear, or piecewise-polynomial function for the pressure loss coefficient:

1. Choose **polynomial**, **piecewise-linear**, or **piecewise-polynomial** in the drop-down list to the right of **Loss Coefficient**. (If the function type you want is already selected, you can click the **Edit...** button to open the dialog box where you will define the function.)

2. In the dialog box that appears for the definition of the **Loss Coefficient** function (for example, Figure 6.59: Polynomial Profile Dialog Box for Loss Coefficient Definition (p. 399)), enter the appropriate values. These profile input dialog boxes are used the same way as the profile input dialog boxes for temperature-dependent properties. See Defining Properties Using Temperature-Dependent Functions (p. 475) to find out how to use them.

Figure 6.59: Polynomial Profile Dialog Box for Loss Coefficient Definition



6.3.20.2.2.2. Constant Value

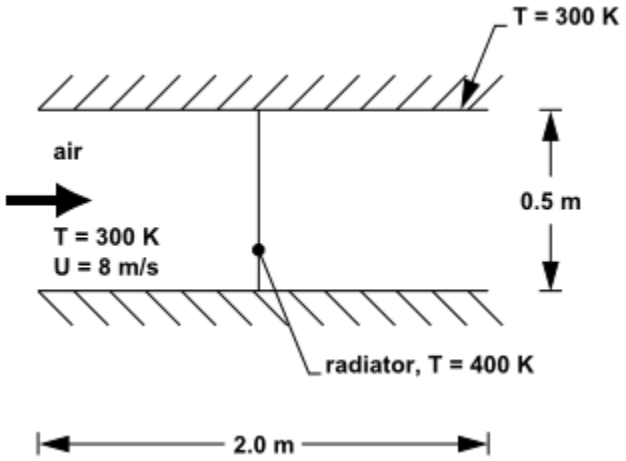
To define a constant loss coefficient, follow these steps:

1. Choose **constant** in the **Loss Coefficient** drop-down list.
2. Enter the value for k_L in the **Loss Coefficient** field.

6.3.20.2.2.3. Example: Calculating the Loss Coefficient

This example shows you how to determine the loss coefficient function. Consider the simple two-dimensional duct flow of air through a water-cooled radiator, shown in Figure 6.60: A Simple Duct with a Radiator (p. 400).

Figure 6.60: A Simple Duct with a Radiator



The radiator characteristics must be known empirically. For this case, assume that the radiator to be modeled yields the test data shown in Table 6.2: Air-side Radiator Data (p. 400), which was taken with a waterside flow rate of 7 kg/min and an inlet water temperature of 400.0 K. To compute the loss coefficient, it is helpful to construct a table with values of the dynamic head, $\frac{1}{2}\rho v^2$, as a function of pressure drop, Δp, and the ratio of these two values, k_L (from Equation 6.130 (p. 396)). (The air density, defined in Figure 6.60: A Simple Duct with a Radiator (p. 400), is 1.0 kg/m^3.) The reduced data are shown in Table 6.3: Reduced Radiator Data (p. 400).

Table 6.2: Air-side Radiator Data

Velocity (m/s)	Upstream Temp (K)	Downstream Temp (K)	Pressure Drop
5.0	300.0	330.0	75.0
10.0	300.0	322.5	250.0
15.0	300.0	320.0	450.0

Table 6.3: Reduced Radiator Data

v (m/s)	$\frac{1}{2}\rho v^2$ (Pa)	Δp (Pa)	k_L
5.0	12.5	75.0	6.0

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

v (m/s)	$\frac{1}{2}\rho v^2$ **(Pa)**	Δp **(Pa)**	k_L
10.0	50.0	250.0	5.0
15.0	112.5	450.0	4.0

The loss coefficient is a linear function of the velocity, decreasing as the velocity increases. The form of this relationship is

$$k_L = 7.0 - 0.2v \tag{6.138}$$

where v is now the absolute value of the velocity through the radiator.

6.3.20.2.3. Defining the Heat Flux Parameters

As mentioned in Radiator Equations (p. 396), you can either define the actual heat flux (q) in the **Heat Flux** field, or set the heat transfer coefficient and radiator temperature (h, T_{ext}). All inputs are in the Radiator Dialog Box (p. 2445).

To define the actual heat flux, specify a **Temperature** of `0` K, and set the constant **Heat Flux** value.

To define the radiator temperature, enter the value for T_{ext} in the **Temperature** field. To define the **Heat-Transfer-Coefficient**, you can specify a polynomial, piecewise-linear, or piecewise-polynomial function of velocity, or a constant value.

6.3.20.2.3.1. Polynomial, Piecewise-Linear, or Piecewise-Polynomial Function

Follow these steps to set a polynomial, piecewise-linear, or piecewise-polynomial function for the heat transfer coefficient:

1. Choose **polynomial**, **piecewise-linear**, or **piecewise-polynomial** in the drop-down list to the right of **Heat-Transfer-Coefficient**. (If the function type you want is already selected, you can click the **Edit...** button to open the dialog box where you will define the function.)

2. In the dialog box that appears for the definition of the **Heat-Transfer-Coefficient** function, enter the appropriate values. These profile input dialog boxes are used the same way as the profile input dialog boxes for temperature-dependent properties. See Defining Properties Using Temperature-Dependent Functions (p. 475) to find out how to use them.

6.3.20.2.3.2. Constant Value

To define a constant heat transfer coefficient, follow these steps:

1. Choose **constant** in the **Heat-Transfer-Coefficient** drop-down list.

2. Enter the value for h in the **Heat-Transfer-Coefficient** field.

6.3.20.2.3.3. Example: Determining the Heat Transfer Coefficient Function

This example shows you how to determine the function for the heat transfer coefficient. Consider the simple two-dimensional duct flow of air through a water-cooled radiator, shown in Figure 6.60: A Simple Duct with a Radiator (p. 400).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The data supplied in Table 6.2: Air-side Radiator Data (p. 400) along with values for the air density (1.0 kg/m^3) and specific heat (1000 J/kg-K) can be used to obtain the following values for the heat transfer coefficient h:

Velocity (m/s)	h **(W/m^2-K)**
5.0	2142.9
10.0	2903.2
15.0	3750.0

The heat transfer coefficient obeys a second-order polynomial relationship (fit to the points in the table above) with the velocity, which is of the form

$$h = 1469.1 + 126.11v + 1.73v^2 \tag{6.139}$$

Note that the velocity v is assumed to be the absolute value of the velocity passing through the radiator.

6.3.20.2.4. Defining Discrete Phase Boundary Conditions for the Radiator

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the radiator. See Setting Boundary Conditions for the Discrete Phase (p. 1301) for details.

6.3.20.3. Postprocessing for Radiators

6.3.20.3.1. Reporting the Radiator Pressure Drop

You can use the Surface Integrals Dialog Box (p. 2616) to report the pressure drop across the radiator, as described in Surface Integration (p. 1920). There are two steps to this procedure:

1. Create a surface on each side of the radiator zone. Use the Transform Surface Dialog Box (p. 2812) (as described in Transforming Surfaces (p. 1732)) to translate the radiator zone slightly upstream and slightly downstream to create two new surfaces.

2. In the **Surface Integrals** dialog box, report the average **Static Pressure** just upstream and just downstream of the radiator. You can then calculate the pressure drop across the radiator.

To check this value against the expected value based on Equation 6.130 (p. 396), you can use the **Surface Integrals** dialog box to report the average normal velocity through the radiator. (If the radiator is not aligned with the x, y, or z axis, you will need to use the Custom Field Function Calculator Dialog Box (p. 2683) to generate a function for the velocity normal to the radiator.) Once you have the average normal velocity, you can use Equation 6.131 (p. 397) to determine the loss coefficient and then Equation 6.130 (p. 396) to calculate the expected pressure loss.

6.3.20.3.2. Reporting Heat Transfer in the Radiator

To determine the temperature rise across the radiator, follow the procedure outlined above for the pressure drop to generate surfaces upstream and downstream of the radiator. Then use the Surface Integrals Dialog Box (p. 2616) (as for the pressure drop report) to report the average **Static Temperature** on each surface. You can then calculate the temperature rise across the radiator.

6.3.20.3.3. Graphical Plots

Graphical reports of interest with radiators are as follows:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Contours or profiles of **Static Pressure** and **Static Temperature**.
- XY plots of **Static Pressure** and **Static Temperature** vs position.

Displaying Graphics (p. 1747) explains how to generate graphical displays of data.

Important

When generating these plots, be sure to turn off the display of node values so that you can see the different values on each side of the radiator. (If you display node values, the cell values on either side of the radiator will be averaged to obtain a node value, and you will not see, for example, the pressure loss across the radiator.)

6.3.21. Porous Jump Boundary Conditions

Porous jump conditions are used to model a thin "membrane" that has known velocity (pressure-drop) characteristics. It is essentially a 1D simplification of the porous media model available for cell zones. Examples of uses for the porous jump condition include modeling pressure drops through screens and filters, and modeling radiators when you are not concerned with heat transfer. This simpler model should be used whenever possible (instead of the full porous media model) because it is more robust and yields better convergence.

The thin porous medium has a finite thickness over which the pressure change is defined as a combination of Darcy's Law and an additional inertial loss term:

$$\Delta p = -\left(\frac{\mu}{\alpha} v + C_2 \frac{1}{2} \rho v^2\right) \Delta m \tag{6.140}$$

where μ is the laminar fluid viscosity, α is the permeability of the medium, C_2 is the pressure-jump coefficient, v is the velocity normal to the porous face, and Δm is the thickness of the medium. Appropriate values for α and C_2 can be calculated using the techniques described in User Inputs for Porous Media (p. 267).

In a coupled analysis involving Fluent and System Coupling, data can be transferred to and from the porous jump boundary. For more about System Coupling, see Performing System Coupling Simulations Using Fluent in Workbench. For more information about the data transferred, see Force transferred to System Coupling from a Porous Jump Boundary.

Note

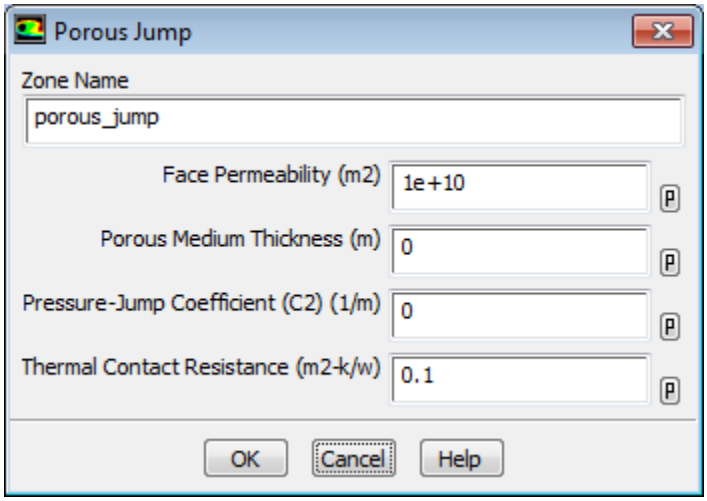
- The porous jump boundary condition considers only the momentum resistance. Diffusion processes are not considered.
- In some cases, the pressure-based coupled solver with porous jump boundary conditions may suffer from convergence issues that do not respond to changes in the coupled solver settings. This behavior depends on the specific flow configuration and porous jump boundary condition values. If convergence instability is observed using the coupled pressure-based solver, it is recommended that you change the pressure-velocity coupling to one of the segregated schemes.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.21.1. User Inputs for the Porous Jump Model

Once the porous jump zone has been identified (in the **Boundary Conditions** task page), you will set all modeling inputs for the porous jump in the Porous Jump Dialog Box (p. 2427) (Figure 6.61: The Porous Jump Dialog Box (p. 404)), which is opened from the Boundary Conditions Task Page (p. 2389) (as described in Setting Cell Zone and Boundary Conditions (p. 241)).

Figure 6.61: The Porous Jump Dialog Box



The inputs required for the porous jump model are as follows:

1. Identify the porous-jump zone.
2. Set the **Face Permeability** of the medium (α in Equation 6.140 (p. 403)).
3. Set the **Porous Medium Thickness** (Δm).
4. Set the **Pressure-Jump Coefficient** (C_2).

> **Note**
>
> The unit for the pressure-jump coefficient (C2) is the inverse of length. Should you want to define different units, you can do so by opening the **Units** dialog box and selecting **length-inverse** from the **Quantities** list.

5. Set the **Thermal Contact Resistance**.
6. Define the discrete phase boundary condition for the porous jump (for discrete phase calculations).
7. If you have enabled the solar load model, define the **Solar Boundary Conditions** for the porous jump. These settings allow you to define the boundary such that it acts as a semi-transparent surface (with respect to the solar radiation calculation only), and therefore allows a portion of the solar radiation to pass through it. See Solar Ray Tracing (p. 900) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.21.1.1. Identifying the Porous Jump Zone

Since the porous jump model is a 1D simplification of the porous media model, the porous-jump zone must be modeled as the interface between cells, rather than a cell zone. Therefore the porous-jump zone is a type of internal face zone (where the faces are line segments in 2D or triangles/quadrilaterals in 3D). If the porous-jump zone is not identified as such by default when you read in the mesh (that is, if it is identified as another type of internal face zone), you can right-click the zone in the tree and use the **Type** sub-menu to change it to a **porous-jump** zone (for more details, see Changing Cell and Boundary Zone Types (p. 239)).

After the zone has been changed to a porous jump, you can open the Porous Jump Dialog Box (p. 2427) (as described in Setting Cell Zone and Boundary Conditions (p. 241)) and specify the porous jump parameters listed above.

6.3.21.1.2. Defining Discrete Phase Boundary Conditions for the Porous Jump

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the porous jump. See Setting Boundary Conditions for the Discrete Phase (p. 1301) for details.

6.3.21.2. Postprocessing for the Porous Jump

Postprocessing suggestions for a problem that includes a porous jump are the same as for porous media problems. See Postprocessing for Porous Media (p. 286).

6.4. Boundary Acoustic Wave Models

The standard boundary conditions, when imposed on the boundaries of an artificially truncated domain, result in the reflection of the outgoing pressure waves. As a consequence, the interior domain will contain spurious wave reflections. Many applications require precise control of the wave reflections from the domain boundaries to obtain accurate flow solutions. ANSYS Fluent offers several boundary wave models that can be used to at the domain boundaries to control these spurious wave reflections.

The following boundary acoustic wave models are available:

- turbo-specific non-reflecting boundary condition (NRBC) (density-based solver only)
- general NRBC (pressure-based and density-based solvers).
- impedance boundary condition (pressure-based solver only)
- transparent flow forcing (pressure-based solver only)

In the density-based solver, the NRBC models are available for flows using the compressible ideal-gas law.

In the pressure-based solver, the general NRBC, impedance, and transient flow forcing models available for transient simulations of compressible flows (including the ideal-gas law, real gas law, species transport, and compressible mixture models).

Information about the available boundary acoustic wave models is provided in the following sections.

6.4.1. Turbo-Specific Non-Reflecting Boundary Conditions
6.4.2. General Non-Reflecting Boundary Conditions
6.4.3. Impedance Boundary Conditions
6.4.4. Transparent Flow Forcing Boundary Conditions

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.4.1. Turbo-Specific Non-Reflecting Boundary Conditions

Information about turbo-specific NRBCs is provided in the following sections.

6.4.1.1. Overview
6.4.1.2. Limitations
6.4.1.3. Theory
6.4.1.4. Using Turbo-Specific Non-Reflecting Boundary Conditions

6.4.1.1. Overview

The standard pressure boundary conditions for compressible flow fix specific flow variables at the boundary (for example, static pressure at an outlet boundary). As a result, pressure waves incident on the boundary will reflect in an unphysical manner, leading to local errors. The effects are more pronounced for internal flow problems where boundaries are usually close to geometry inside the domain, such as compressor or turbine blade rows.

The turbo-specific non-reflecting boundary conditions permit waves to "pass" through the boundaries without spurious reflections. The method used in ANSYS Fluent is based on the Fourier transformation of solution variables at the non-reflecting boundary [36] (p. 2888). Similar implementations have been investigated by other authors [72] (p. 2890) [97] (p. 2892). The solution is rearranged as a sum of terms corresponding to different frequencies, and their contributions are calculated independently. While the method was originally designed for axial turbomachinery, it has been extended for use with radial turbomachinery.

6.4.1.2. Limitations

Note the following limitations of turbo-specific NRBCs:

- They are available only with the density-based solver (explicit or implicit).
- The current implementation applies to steady compressible flows, with the density calculated using the ideal gas law.
- Inlet and outlet boundary conditions must be pressure inlets and outlets only.

  Important

  Note that the pressure inlet boundaries must be set to the cylindrical coordinate flow specification method when turbo-specific NRBCs are used.

- Quad-mapped (structured) surface meshes must be used for inflow and outflow boundaries in a 3D geometry (that is, triangular or quad-paved surface meshes are not allowed). See Figure 6.62: Mesh and Prescribed Boundary Conditions in a 3D Axial Flow Problem (p. 407) and Figure 6.63: Mesh and Prescribed Boundary Conditions in a 3D Radial Flow Problem (p. 408) for examples.

  Important

  Note that you may use unstructured meshes in 2D geometries (Figure 6.64: Mesh and Prescribed Boundary Conditions in a 2D Case (p. 409)), and an unstructured mesh may be used away from the inlet and outlet boundaries in 3D geometries.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The turbo-specific NRBC [36] (p. 2888) has been extended for use on 3D geometries [97] (p. 2892) by decoupling the tangential flow variations from the radial variations. This approximation works best for geometries with a blade pitch that is small compared to the radius of the geometry.
- Reverse flow on the inflow and outflow boundaries are not allowed. If strong reverse flow is present, then you should consider using the General NRBCs instead.
- NRBCs are not compatible with species transport models. They are mainly used to solve ideal-gas single-species flow.

Figure 6.62: Mesh and Prescribed Boundary Conditions in a 3D Axial Flow Problem

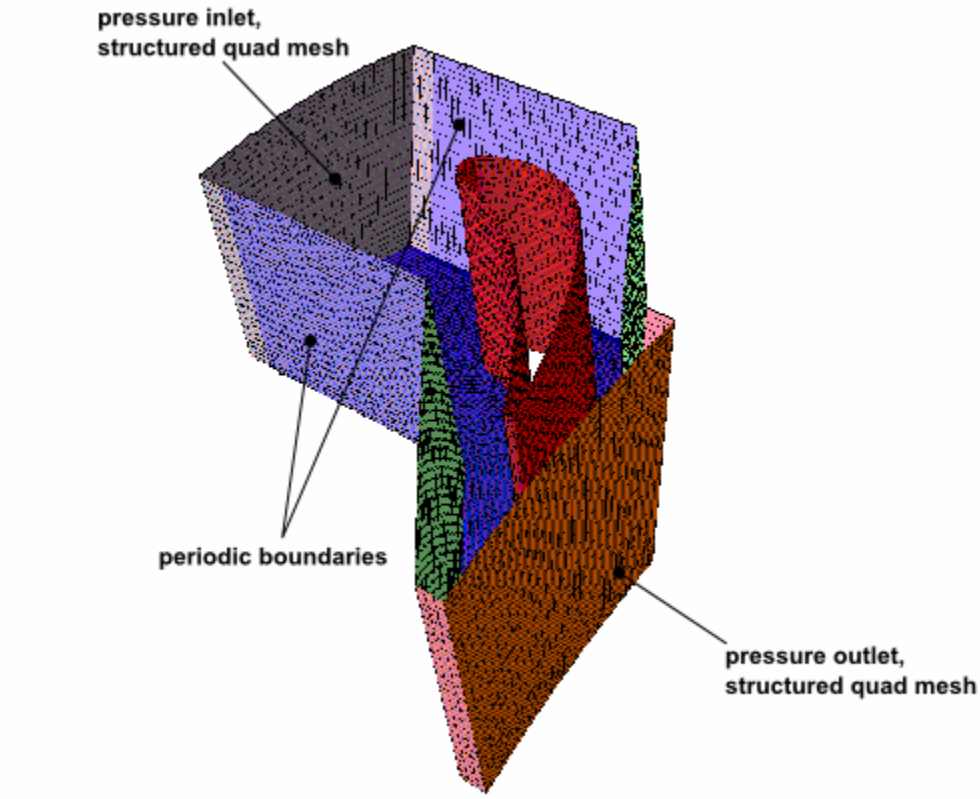
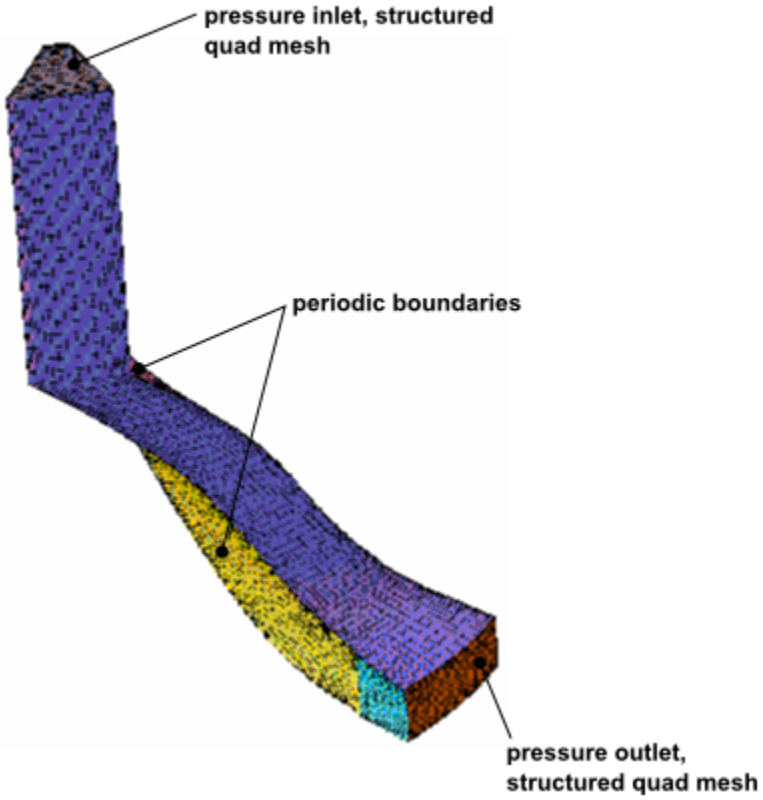
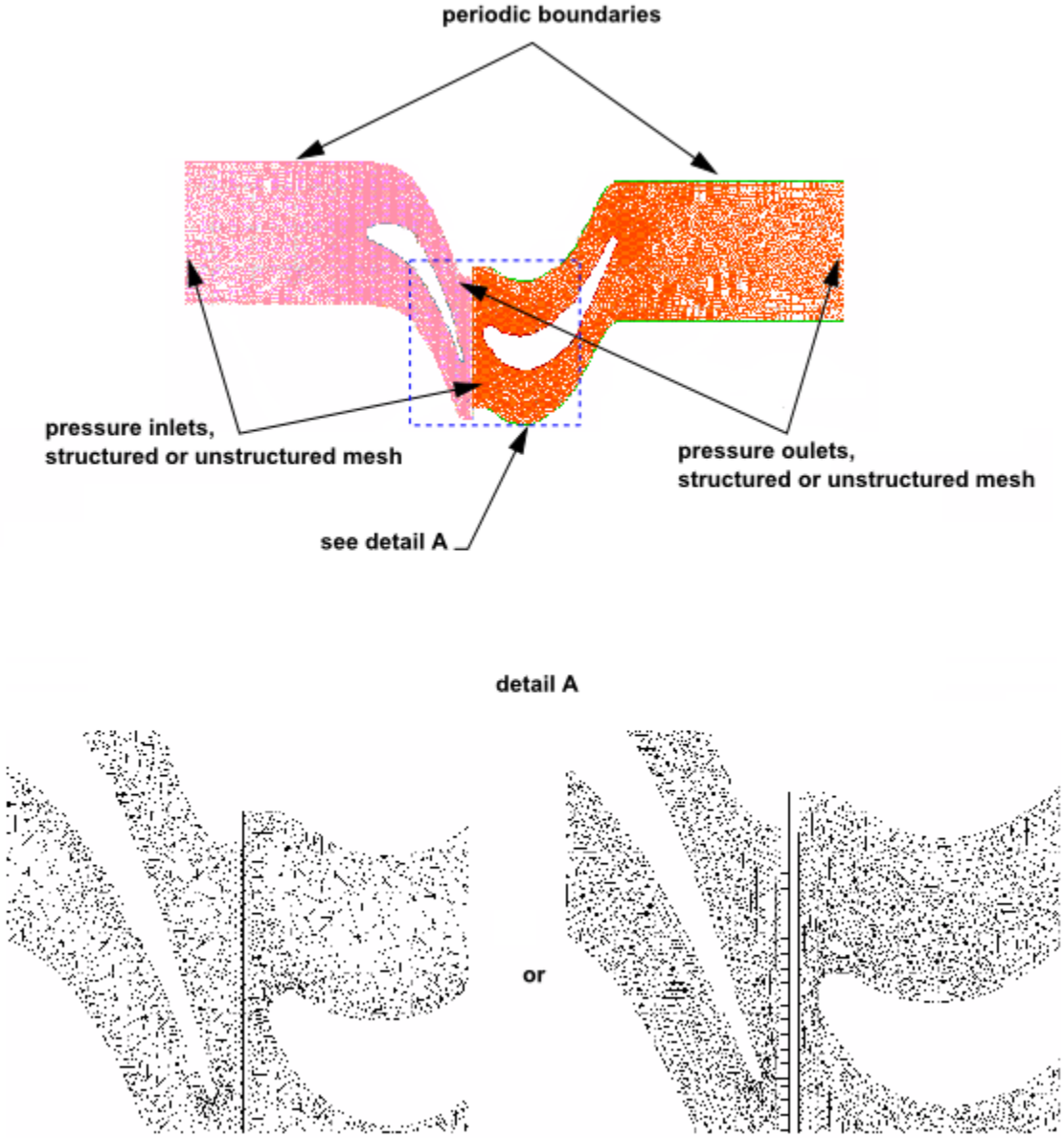


Figure 6.63: Mesh and Prescribed Boundary Conditions in a 3D Radial Flow Problem



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.64: Mesh and Prescribed Boundary Conditions in a 2D Case



6.4.1.3. Theory

Turbo-specific NRBCs are based on Fourier decomposition of solutions to the linearized Euler equations. The solution at the inlet and outlet boundaries is circumferentially decomposed into Fourier modes, with the 0th mode representing the average boundary value (which is to be imposed as a user input), and higher harmonics that are modified to eliminate reflections [97] (p. 2892).

6.4.1.3.1. Equations in Characteristic Variable Form

In order to treat individual waves, the linearized Euler equations are transformed to characteristic variable (C_i) form. If we first consider the 1D form of the linearized Euler equations, it can be shown that the characteristic variables C_i are related to the solution variables as follows:

$$\tilde{Q} = T^{-1} C \tag{6.141}$$

where

$$\tilde{Q}=\begin{Bmatrix}\tilde{\rho}\\ \tilde{u}_a\\ \tilde{u}_t\\ \tilde{u}_r\\ \tilde{p}\end{Bmatrix},T^{-1}=\begin{bmatrix}-\frac{1}{\overline{a}^2} & 0 & 0 & \frac{1}{2\overline{a}^2} & \frac{1}{2\overline{a}^2}\\ 0 & 0 & 0 & \frac{1}{2\overline{\rho}\,\overline{a}} & \frac{1}{2\overline{\rho}\,\overline{a}}\\ 0 & \frac{1}{\overline{\rho}\,\overline{a}} & 0 & 0 & 0\\ 0 & 0 & \frac{1}{\overline{\rho}\,\overline{a}} & 0 & 0\\ 0 & 0 & 0 & \frac{1}{2} & \frac{1}{2}\end{bmatrix},C=\begin{Bmatrix}C_1\\ C_2\\ C_3\\ C_4\\ C_5\end{Bmatrix} \tag{6.142}$$

where $\overline{a}$ is the average acoustic speed along a boundary zone, $\tilde{\rho}$, $\tilde{u}_a$, $\tilde{u}_t$, $\tilde{u}_r$, and $\tilde{p}$ represent perturbations from a uniform condition (for example, $\tilde{\rho}=\rho-\overline{\rho}$,$\tilde{p}=p-\overline{p}$, and so on).

Note that the analysis is performed using the cylindrical coordinate system. All overlined (averaged) flow field variables (for example, $\overline{\rho}$, $\overline{a}$) are intended to be averaged along the pitchwise direction.

In quasi-3D approaches [36] (p. 2888) [72] (p. 2890) [97] (p. 2892), a procedure is developed to determine the changes in the characteristic variables, denoted by δC_i, at the boundaries such that waves will not reflect. These changes in characteristic variables are determined as follows:

$$\delta C=T\,\delta Q \tag{6.143}$$

where

$$\delta C=\begin{Bmatrix}\delta C_1\\ \delta C_2\\ \delta C_3\\ \delta C_4\\ \delta C_5\end{Bmatrix},T=\begin{bmatrix}-\overline{a}^2 & 0 & 0 & 0 & 1\\ 0 & 0 & \overline{\rho}\,\overline{a} & 0 & 0\\ 0 & 0 & 0 & \overline{\rho}\,\overline{a} & 0\\ 0 & \overline{\rho}\,\overline{a} & 0 & 0 & 1\\ 0 & -\overline{\rho}\,\overline{a} & 0 & 0 & 1\end{bmatrix},\delta Q=\begin{Bmatrix}\delta\rho\\ \delta u_a\\ \delta u_t\\ \delta u_r\\ \delta p\end{Bmatrix} \tag{6.144}$$

The changes to the outgoing characteristics — one characteristic for subsonic inflow (δC_5), and four characteristics for subsonic outflow (δC_1, δC_2, δC_3, δC_4) — are determined from extrapolation of the flow field variables using Equation 6.143 (p. 410).

The changes in the incoming characteristics — four characteristics for subsonic inflow (δC_1, δC_2, δC_3, δC_4), and one characteristic for subsonic outflow (δC_5) — are split into two components: average change along the boundary ($\delta\overline{C}_i$), and local changes in the characteristic variable due to harmonic variation along the boundary (δC_{iL}). The incoming characteristics are therefore given by

$$\delta C_{i_j}=\delta C_{iold_j}+\sigma\left(\delta C_{inew_j}-\delta C_{iold_j}\right) \tag{6.145}$$

$$\delta C_{inew_j}=\left(\delta\overline{C}_i+\delta C_{iL_j}\right) \tag{6.146}$$

where $i=1,2,3,4$ on the inlet boundary or $i=5$ on the outlet boundary, and $j=1,\ldots,N$ is the grid index in the pitchwise direction including the periodic point once. The under-relaxation factor σ has a default value of 0.75. Note that this method assumes a periodic solution in the pitchwise direction.

The flow is decomposed into mean and circumferential components using Fourier decomposition. The 0th Fourier mode corresponds to the average circumferential solution, and is treated according to the standard 1D characteristic theory. The remaining parts of the solution are described by a sum of harmonics, and treated as 2D non-reflecting boundary conditions [36] (p. 2888).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.4.1.3.2. Inlet Boundary

For subsonic inflow, there is one outgoing characteristic (δC_5) determined from Equation 6.143 (p. 410), and four incoming characteristics (δC_1, δC_2, δC_3, δC_4) calculated using Equation 6.145 (p. 410). The average changes in the incoming characteristics are computed from the requirement that the entropy (s), radial and tangential flow angles (α_r and α_t), and stagnation enthalpy (h_0) are specified. Note that in ANSYS Fluent you can specify p_0 and T_0 at the inlet, from which s_{in} and $h_{0_{in}}$ are easily obtained. This is equivalent to forcing the following four residuals to be zero:

$$R_1 = \overline{\rho}\left(\overline{s} - s_{in}\right) \tag{6.147}$$

$$R_2 = \overline{\rho}\,\overline{a}\left(\overline{u}_t - \overline{u}_a \tan\alpha_t\right) \tag{6.148}$$

$$R_3 = \overline{\rho}\,\overline{a}\left(\overline{u}_r - \overline{u}_a \tan\alpha_r\right) \tag{6.149}$$

$$R_4 = \overline{\rho}\left(\overline{h}_0 - h_{0_{in}}\right) \tag{6.150}$$

where

$$s_{in} = \gamma \ln\left(T_{0_{in}}\right) - (\gamma - 1)\ln\left(p_{0_{in}}\right) \tag{6.151}$$

$$h_{0_{in}} = c_p T_{0_{in}} \tag{6.152}$$

The average characteristic is then obtained from residual linearization as follows (see also Figure 6.65: Prescribed Inlet Angles (p. 412)

$$\begin{Bmatrix} \delta\overline{C}_1 \\ \delta\overline{C}_2 \\ \delta\overline{C}_3 \\ \delta\overline{C}_4 \end{Bmatrix} = \begin{bmatrix} -1 & 0 & 0 & 0 \\ \frac{\tan\alpha_t}{(1-\gamma)M} & \frac{M_1}{M} & \frac{-M_t}{M}\tan\alpha_t & \frac{\tan\alpha_t}{M} \\ \frac{\tan\alpha_r}{(\gamma-1)M} & \frac{M_t}{M}\tan\alpha_t & \frac{M_2}{M} & \frac{-\tan\alpha_r}{M} \\ \frac{2}{(\gamma-1)M} & 2\frac{M_t}{M} & 2\frac{M_r}{M} & \frac{-2}{M} \end{bmatrix} \begin{Bmatrix} R_1 \\ R_2 \\ R_3 \\ R_4 \end{Bmatrix} \tag{6.153}$$

where

$$M_a = \frac{\overline{u}_a}{\overline{a}} \tag{6.154}$$

$$M_t = \frac{\overline{u}_t}{\overline{a}} \tag{6.155}$$

$$M_r = \frac{\overline{u}_r}{\overline{a}} \tag{6.156}$$

and

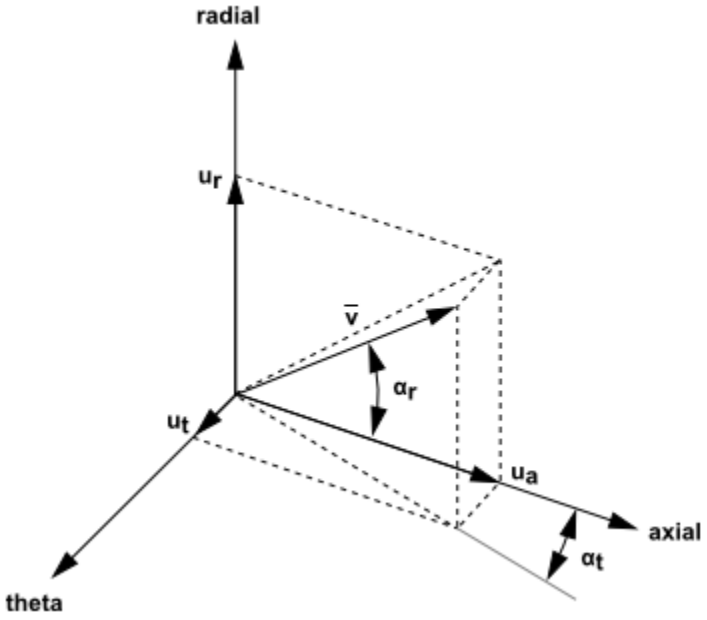
$$M = 1 + M_a - M_t\tan\alpha_t + M_r\tan\alpha_r \tag{6.157}$$

$$M_1 = -1 - M_a - M_r\tan\alpha_r \tag{6.158}$$

$$M_2 = -1 - M_a - M_t\tan\alpha_t \tag{6.159}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.65: Prescribed Inlet Angles



where

$$|v|=\sqrt{u_t^2+u_r^2+u_a^2} \tag{6.160}$$

$$e_t=\frac{u_t}{|v|} \tag{6.161}$$

$$e_r=\frac{u_r}{|v|} \tag{6.162}$$

$$e_a=\frac{u_a}{|v|} \tag{6.163}$$

$$\tan\alpha_t=\frac{e_t}{e_a} \tag{6.164}$$

$$\tan\alpha_r=\frac{e_r}{e_a} \tag{6.165}$$

To address the local characteristic changes at each j grid point along the inflow boundary, the following relations are developed [36] (p. 2888) [97] (p. 2892):

$$\begin{aligned}
\delta C_{1L_j}&=\overline{p}\left(s_j-\overline{s}\right)\\
\delta C_{2L_j}&=C'_{2_j}-\overline{\rho a}\left(u_{t_j}-\overline{u}_t\right)\\
\delta C_{3L_j}&=-\overline{\rho a}\left(u_{r_j}-\overline{u}_r\right)\\
\delta C_{4L_j}&=\frac{-2}{(1+M_{aj})}\left(\frac{1}{\gamma-1}\delta C_{1L_j}+M_{t_j}\delta C_{2L_j}+M_{r_j}\delta C_{3L_j}+\overline{\rho}\left(h_{0_j}-\overline{h}_0\right)\right)
\end{aligned} \tag{6.166}$$

Note that the relation for the first and fourth local characteristics force the local entropy and stagnation enthalpy to match their average steady-state values.

The characteristic variable C'_{2_j} is computed from the inverse discrete Fourier transform of the second characteristic. The discrete Fourier transform of the second characteristic in turn is related to the discrete Fourier transform of the fifth characteristic. Hence, the characteristic variable C'_{2_j} is computed along the pitch as follows:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$C_{2_j}^{'}=2\Re\left(\sum_{n=1}^{\frac{N}{2}-1}\hat{C}_{2_n}\exp\left(i2\pi n\frac{\theta_j-\theta_1}{\theta_N-\theta_1}\right)\right) \tag{6.167}$$

The Fourier coefficients $C_{2_n}^{'}$ are related to a set of equidistant distributed characteristic variables $C_{5_j}^{*}$ by the following [72] (p. 2890):

$$\hat{C}_{2_n}=\begin{cases}-\frac{\overline{u}_t+B}{N(\overline{a}+\overline{u}_a)}\sum_{j=1}^{N}C_{5_j}^{*}\exp\left(-i2\pi\frac{jn}{N}\right) & \beta>0\\ -\frac{\overline{u}_t+B}{\overline{a}+\overline{u}_a}C_{5_j} & \beta<0\end{cases} \tag{6.168}$$

where

$$B=\begin{cases}i\sqrt{\beta} & \beta>0\\ -sign(\overline{u}_t)\sqrt{|\beta|} & \beta<0\end{cases} \tag{6.169}$$

and

$$\beta=\overline{a}^2-\overline{u}_a^2-\overline{u}_t^2 \tag{6.170}$$

The set of equidistributed characteristic variables $C_{5_j}^{*}$ is computed from arbitrary distributed C_{5_j} by using a cubic spline for interpolation, where

$$C_{5_j}=-\overline{\rho}\,\overline{a}\left(u_{a_j}-\overline{u}_a\right)+\left(p_j-\overline{p}\right) \tag{6.171}$$

For supersonic inflow the user-prescribed static pressure ($p_{s_{in}}$) along with total pressure ($p_{0_{in}}$) and total temperature ($T_{0_{in}}$) are sufficient for determining the flow condition at the inlet.

6.4.1.3.3. Outlet Boundary

For subsonic outflow, there are four outgoing characteristics (δC_1, δC_2, δC_3, and δC_4) calculated using Equation 6.143 (p. 410), and one incoming characteristic (δC_5) determined from Equation 6.145 (p. 410). The average change in the incoming fifth characteristic is given by

$$\delta\overline{C}_5=-2\left(\overline{p}-p_{out}\right) \tag{6.172}$$

where $\overline{p}$ is the current averaged pressure at the exit plane and p_{out} is the desirable average exit pressure (this value is specified by you for single-blade calculations or obtained from the assigned profile for mixing-plane calculations). The local changes (δC_{5L_j}) are given by

$$\delta C_{5L_j}=C_{5_j}^{'}+\overline{\rho}\,\overline{a}\left(u_{a_j}-\overline{u}_a\right)-\left(p_j-\overline{p}\right) \tag{6.173}$$

The characteristic variable $C_{5_j}^{'}$ is computed along the pitch as follows:

$$C_{5_j}^{'}=2\Re\left(\sum_{n=1}^{\frac{N}{2}-1}\hat{C}_{5_n}\exp\left(i2\pi n\frac{\theta_j-\theta_1}{\theta_N-\theta_1}\right)\right) \tag{6.174}$$

The Fourier coefficients $\hat{C}_{5_n}$ are related to two sets of equidistantly distributed characteristic variables ($C_{2_j}^{*}$ and $C_{4_j}^{*}$, respectively) and given by the following [72] (p. 2890):

$$\hat{C}_{5_n} = \begin{cases} \frac{A_2}{N}\sum_{j=1}^{N} C^*_{2_j} \exp\left(i2\pi\frac{jn}{N}\right) - \frac{A_4}{N}\sum_{j=1}^{N} C^*_{4_j} \exp\left(i2\pi\frac{jn}{N}\right) & \beta > 0 \\ A_2 C_{2_j} - A_4 C_{4_j} & \beta < 0 \end{cases} \tag{6.175}$$

where

$$A_2 = \frac{2\overline{u}_a}{B - \overline{u}_t} \tag{6.176}$$

$$A_4 = \frac{B + \overline{u}_t}{B - \overline{u}_t} \tag{6.177}$$

The two sets of equidistributed characteristic variables ($C^*_{2_j}$ and $C^*_{4_j}$) are computed from arbitrarily distributed C_{2_j} and C_{4_j} characteristics by using a cubic spline for interpolation, where

$$C_{2_j} = \overline{\rho}\,\overline{a}\left(u_{t_j} - \overline{u}_t\right) \tag{6.178}$$

$$C_{4_j} = \overline{\rho}\,\overline{a}\left(u_{a_j} - \overline{u}_a\right) + \left(p_j - \overline{p}\right) \tag{6.179}$$

For supersonic outflow all flow field variables are extrapolated from the interior.

6.4.1.3.4. Updated Flow Variables

Once the changes in the characteristics are determined on the inflow or outflow boundaries, the changes in the flow variables δQ can be obtained from Equation 6.143 (p. 410). Therefore, the values of the flow variables at the boundary faces are as follows:

$$p_f = p_j + \delta p \tag{6.180}$$

$$u_{a_f} = u_{a_j} + \delta u_a \tag{6.181}$$

$$u_{t_f} = u_{t_j} + \delta u_t \tag{6.182}$$

$$u_{r_f} = u_{r_j} + \delta u_r \tag{6.183}$$

$$T_f = T_j + \delta T \tag{6.184}$$

6.4.1.4. Using Turbo-Specific Non-Reflecting Boundary Conditions

Important

If you intend to use turbo-specific NRBCs in conjunction with the density-based implicit solver, it is recommended that you first converge the solution before turning on turbo-specific NRBCs, then converge it again with turbo-specific NRBCs turned on. If the solution is diverging, then you should lower the CFL number. These steps are necessary because only approximate flux Jacobians are used for the pressure-inlet and pressure-outlet boundaries when turbo-specific NRBCs are activated with the density-based implicit solver.

The procedure for using the turbo-specific NRBCs is as follows:

1. Turn on the turbo-specific NRBCs using the `non-reflecting-bc` text command:

   `define` → `boundary-conditions` → `non-reflecting-bc` → `turbo-specific-nrbc` → `enable?`

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you are not sure whether or not NRBCs are turned on, use the `show-status` text command.

2. Perform NRBC initialization using the `initialize` text command:

   `define` → `boundary-conditions` → `non-reflecting-bc` → `turbo-specific-nrbc` → `initialize`

   If the initialization is successful, a summary printout of the domain extent will be displayed. If the initialization is not successful, an error message will be displayed indicating the source of the problem. The initialization will set up the pressure-inlet and pressure-outlet boundaries for use with turbo-specific NRBCs.

   Important

   Note that the pressure inlet boundaries must be set to the cylindrical coordinate flow specification method when turbo-specific NRBCs are used.

3. If necessary, modify the parameters in the `set/` submenu:

   `define` → `boundary-conditions` → `non-reflecting-bc` → `turbo-specific-nrbc` → `set`

   `under-relaxation`
   allows you to set the value of the under-relaxation factor σ in Equation 6.145 (p. 410). The default value is 0.75.

   `discretization`
   allows you to set the discretization scheme. The default is to use higher-order reconstruction if available.

   `verbosity`
   allows you to control the amount of information printed to the console during an NRBC calculation.

   - `0` : silent
   - `1` : basic information (default)
   - 2: detailed information (for debugging purposes only)

6.4.1.4.1. Using the NRBCs with the Mixing-Plane Model

If you want to use the NRBCs with the mixing-plane model you must define the mixing plane interfaces as pressure-outlet and pressure-inlet zone type pairs.

Important

Turbo-specific NRBCs should not be used with the mixing-plane model if reverse flow is present across the mixing-plane.

6.4.1.4.2. Using the NRBCs in Parallel ANSYS Fluent

When the turbo-specific NRBCs are used in conjunction with the parallel solver, all cells in each boundary zone, where NRBCs will be applied, must be located or contained within a single partition.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can ensure this by manually partitioning the mesh (see Partitioning the Mesh Manually and Balancing the Load (p. 2033) for more information).

6.4.2. General Non-Reflecting Boundary Conditions

Information about general NRBCs is provided in the following sections.

6.4.2.1. Overview
6.4.2.2. Restrictions and Limitations
6.4.2.3. Theory
6.4.2.4. Using the General Non-Reflecting Boundary Condition

6.4.2.1. Overview

The general non-reflecting boundary conditions in ANSYS Fluent are based on characteristic wave relations derived from the Euler equations. In the density-based solver, the non-reflecting boundary conditions are applied only on pressure-outlet boundary conditions. In the pressure-based solver they are applied on pressure-inlet, pressure-outlet, velocity-inlet and mass-flux boundary conditions. To obtain the primitive flow quantities (P,u,v,w,T), reformulated Euler equations are solved on the boundary of the domain in an algorithm similar to the flow equations applied to the interior of the domain.

Unlike the turbo-specific NRBC, the general NRBC method is not restricted by geometric constraints or mesh type. However, good cell skewness near the boundaries where the NRBCs can be applied for better convergence.

6.4.2.2. Restrictions and Limitations

Note the following restrictions and limitations on the general NRBCs:

- The general NRBC is not available if the target mass flow rate is activated in the pressure-outlet dialog box.
- The general NRBC using the density-based solver is available only with compressible flow while using the ideal-gas law.

> **Important**
>
> The general NRBC using the density-based solver should not be used with the wet steam or real gas models.

- The general NRBCs using the density-based solver are not compatible with species transport and mixture fraction transport models (for premixed and partially-premixed models). They are mainly used to solve ideal-gas single-component flow.
- The general NRBCs using the pressure-based solver are not compatible with steady-state cases, all multiphase models, and compressible liquids models.

6.4.2.3. Theory

General NRBCs are derived by first recasting the Euler equations in an orthogonal coordinate system (x_1,x_2,x_3) such that one of the coordinates, x_1, is normal to the boundary Figure 6.66: The Local Orthogonal Coordinate System onto which Euler Equations are Recasted for the General NRBC Method (p. 418). The characteristic analysis [123] (p. 2893) [124] (p. 2893) is then used to modify terms corresponding to

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

waves propagating in the x_1 normal direction. When doing so, a system of equations can be written to describe the wave propagation as follows:

$$\begin{aligned}
&\frac{\partial \rho}{\partial t} + d_1 + \frac{\partial m_2}{\partial x_2} + \frac{\partial m_3}{\partial x_3} = 0 \\
&\frac{\partial m_1}{\partial t} + U_1 d_1 + \rho d_3 + \frac{\partial (m_1 U_2)}{\partial x_2} + \frac{\partial (m_1 U_3)}{\partial x_3} = 0 \\
&\frac{\partial m_2}{\partial t} + U_2 d_1 + \rho d_4 + \frac{\partial (m_2 U_2)}{\partial x_2} + \frac{\partial (m_2 U_3)}{\partial x_3} + \frac{\partial P}{\partial x_2} = 0 \\
&\frac{\partial m_3}{\partial t} + U_3 d_1 + \rho d_5 + \frac{\partial (m_3 U_2)}{\partial x_2} + \frac{\partial (m_3 U_3)}{\partial x_3} + \frac{\partial P}{\partial x_3} = 0 \\
&\frac{\partial \rho E}{\partial t} + \frac{1}{2}|V|^2 d_1 + \frac{d_2}{(\gamma - 1)} + m_1 d_3 + m_2 d_4 + m_3 d_5 + \frac{\partial [(\rho E + P) U_2]}{\partial x_2} + \frac{\partial [(\rho E + P) U_3]}{\partial x_3} = 0
\end{aligned} \tag{6.185}$$

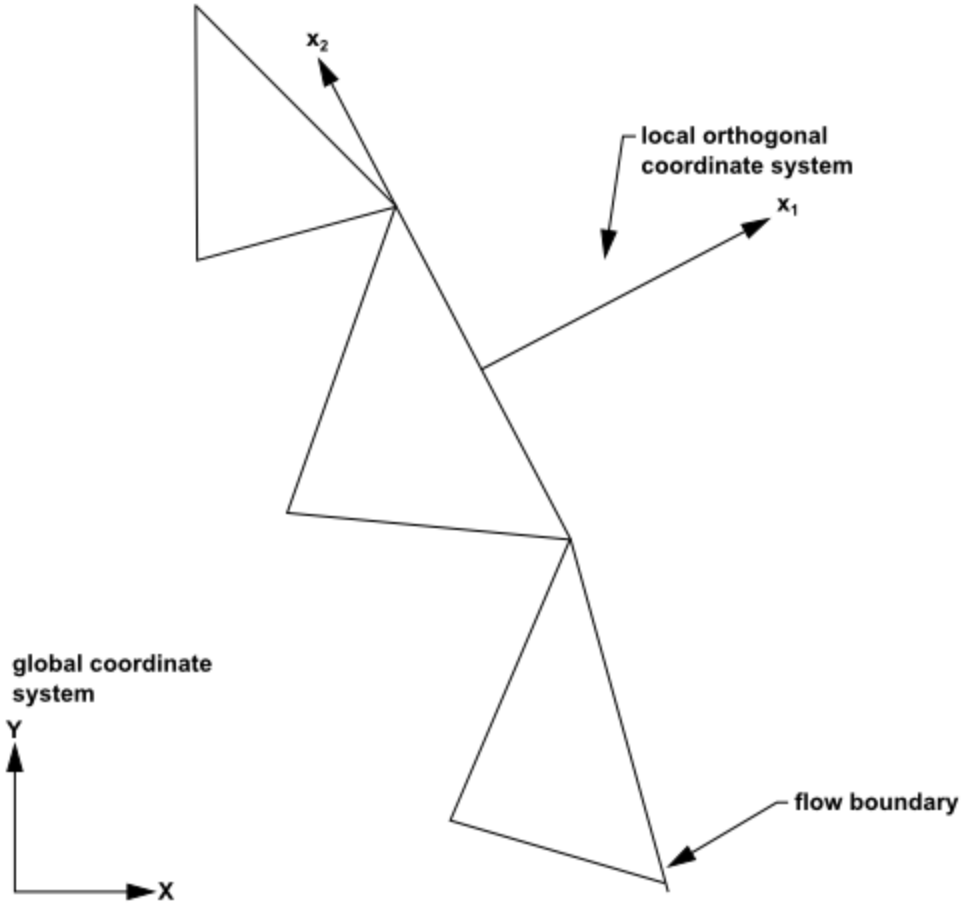
Where $m_1 = \rho U_1$, $m_2 = \rho U_2$ and $m_3 = \rho U_3$ and U_1, U_2 and U_3 are the velocity components in the coordinate system (x_1, x_2, x_3). The equations above are solved on non-reflecting boundaries, along with the interior governing flow equations, using similar time stepping algorithms to obtain the values of the primitive flow variables (P, u, v, w, T).

Important

Note that a transformation between the local orthogonal coordinate system (x_1, x_2, x_3) and the global Cartesian system (X, Y, Z) must be defined on each face on the boundary to obtain the velocity components (u, v, w) in a global Cartesian system.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.66: The Local Orthogonal Coordinate System onto which Euler Equations are Recasted for the General NRBC Method



The d_i terms in the transformed Euler equations contain the outgoing and incoming characteristic wave amplitudes, L_i, and are defined as follows:

$$\begin{aligned} d_1 &= \frac{1}{c^2}\left[L_2 + \frac{1}{2}\left(L_5 + L_1\right)\right] \\ d_2 &= \frac{1}{2}\left(L_5 + L_1\right) \\ d_3 &= \frac{1}{2\rho c}\left(L_5 - L_1\right) \\ d_4 &= L_3 \\ d_5 &= L_4 \end{aligned} \tag{6.186}$$

From characteristic analyses, the wave amplitudes, L_i, are given by:

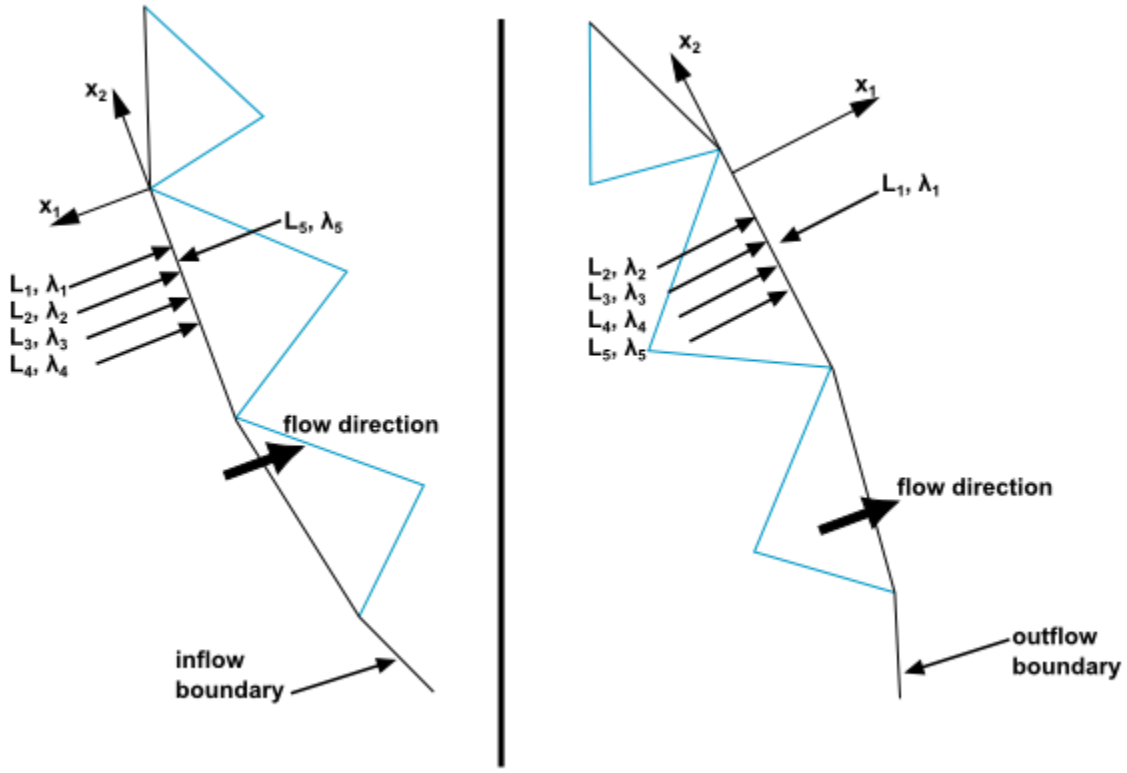
$$\begin{aligned} L_1 &= \lambda_1\left(\frac{\partial P}{\partial x_1} - \rho c \frac{\partial U_1}{\partial x_1}\right) \\ L_2 &= \lambda_2\left(c^2 \frac{\partial \rho}{\partial x_1} - \frac{\partial P}{\partial x_1}\right) \\ L_3 &= \lambda_3 \frac{\partial U_2}{\partial x_1} \\ L_4 &= \lambda_4 \frac{\partial U_3}{\partial x_1} \\ L_5 &= \lambda_5\left(\frac{\partial P}{\partial x_1} + \rho c \frac{\partial U_1}{\partial x_1}\right) \end{aligned} \tag{6.187}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The outgoing and incoming characteristic waves are associated with the characteristic velocities of the system (i.e eigenvalues), λ_i, as seen in Figure 6.67: Waves Leaving and Entering a Boundary Face on Inflow and Outflow Boundaries. The Wave Amplitudes are Shown with the Associated Eigenvalues for a Subsonic Flow Condition (p. 419). These eigenvalues are given by:

$$\begin{aligned}\lambda_1&=U_1-c\\ \lambda_2&=\lambda_3=\lambda_4=U_1\\ \lambda_5&=U_1+c\end{aligned} \tag{6.188}$$

Figure 6.67: Waves Leaving and Entering a Boundary Face on Inflow and Outflow Boundaries. The Wave Amplitudes are Shown with the Associated Eigenvalues for a Subsonic Flow Condition



For subsonic flow leaving a boundary, four waves leave the domain (associated with positive eigenvalues λ_2, λ_3, λ_4, and λ_5) and one enters the domain (associated with negative eigenvalue λ_1). For subsonic flow entering a boundary, four waves enter the domain (associated with the negative eigenvalues λ_1, λ_2, λ_3, λ_4) and one leaves the domain (associated with a positive eigenvalue λ_5).

To solve Equation 6.185 (p. 417) on a boundary, the amplitude of the incoming and outgoing waves must be first determined. The amplitude of outgoing waves are computed from Equation 6.187 (p. 418) by using extrapolated values of flow derivatives $\frac{\partial P}{\partial x_1}$, $\frac{\partial \rho}{\partial x_1}$, $\frac{\partial U_1}{\partial x_1}$, $\frac{\partial U_2}{\partial x_1}$, and $\frac{\partial U_3}{\partial x_1}$ from inside the domain. The amplitude of the incoming waves are computed as follows. The amplitude of waves L_3, L_4, for tangential velocity components, is set to zero. The amplitude of the incoming pressure and entropy waves are computed from the Linear Relaxation Method (LRM) of Poinsot [85] (p. 2891) [86] (p. 2891). The LRM method sets the value of the incoming wave amplitude to be proportional to the differences between the local primitive variable on a boundary face and the imposed boundary value. The expression for the amplitudes depends on the boundary type as shown below:

1. **Pressure Outlet**

$$L_1=K(P-P_{exit}) \tag{6.189}$$

where P_{exit} is the imposed pressure at the exit boundary, K is the relaxation factor, and P is the local pressure value at the boundary.

In general, the desirable average pressure on a non-reflecting boundary can be either relaxed toward a pressure value at infinity or enforced to be equivalent to some desired pressure at the exit of the boundary.

In the case of a backflow, the pressure-based solver has two options for specifying pressure at the boundary; static pressure or total pressure. If the static pressure option is selected, the pressure at infinity is equal to the pressure defined in the boundary dialog box. If the total pressure option is selected, the pressure at infinity is a total pressure that is computed from the value in the dialog box. For the density-based solver, the total pressure option is used and no other options are available.

If you want the average pressure at the boundary to relax toward P at infinity (that is $P_{exit}=P_{\infty}$), the suggested K factor is given by:

$$K=\sigma_1\left(1-M_{max}{}^2\right)\frac{c}{h} \tag{6.190}$$

where c is the acoustic speed, h is the domain size, M_{max} is the maximum Mach number in the domain, and σ_1 is the under-relaxation factor (default value is 0.15). On the other hand, if the desired average pressure at the boundary is to approach a specific imposed value at the boundary, then the K factor is given by:

$$K=\sigma_2 c \tag{6.191}$$

where the default value for σ_2 is 5.0

Important

The desired average pressure option is only available for the density-based solver.

2. **Pressure Inlet**:

$$\begin{aligned} L_1&=K\left(P-P_{exit}\right)\\ L_2&=K\left[c^2\left(\rho-\rho_{exit}\right)-\left(P-P_{exit}\right)\right] \end{aligned} \tag{6.192}$$

where the imposed pressure, P_{exit}, and density, ρ_{exit}, are computed from the specified total pressure and total temperature.

3. **Mass Flux Boundary**:

$$\begin{aligned} L_1&=-K\rho c\left(U_1-U_{1,exit}\right)\\ L_2&=Kc^2\left(\rho-\rho_{exit}\right) \end{aligned} \tag{6.193}$$

where the imposed exit velocity and density are computed from specific mass flux and total temperature.

4. **Velocity Inlet**:

$$\begin{aligned} L_1&=-K\rho c\left(U_1-U_{1,exit}\right)\\ L_2&=Kc^2\left(\rho-\rho_{exit}\right) \end{aligned} \tag{6.194}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where the imposed velocity, $U_{1,exit}$, is specified at the boundary and the imposed density is computed from the specified boundary temperature and extrapolated pressure.

6.4.2.4. Using the General Non-Reflecting Boundary Condition

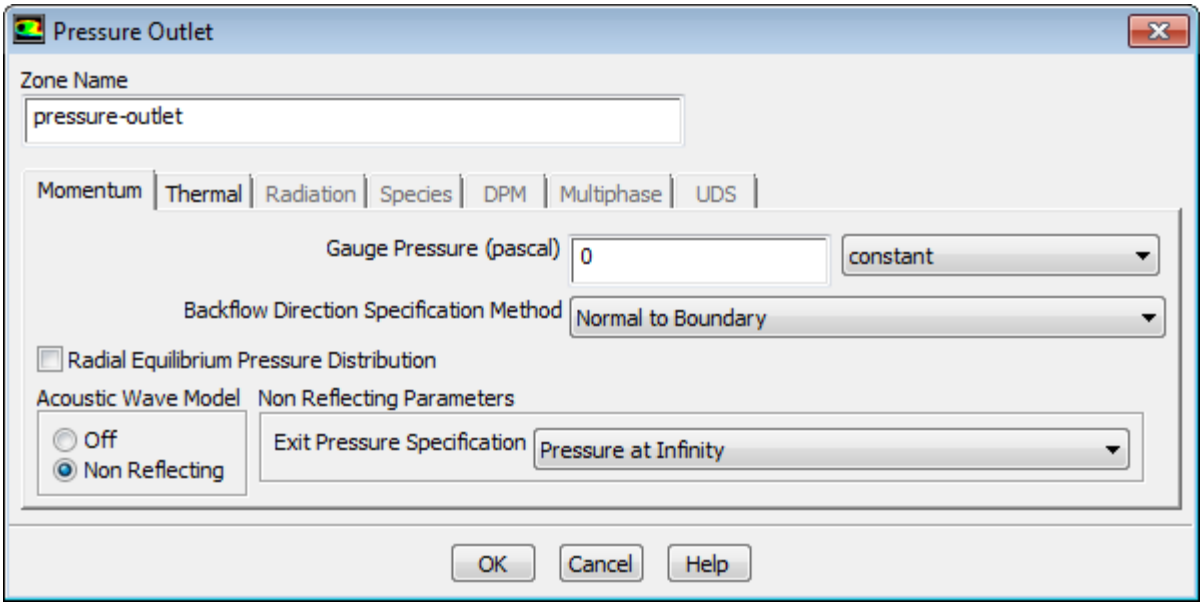
The general NRBC is available for use in the **Pressure Outlet** dialog box when either the density-based (with ideal gas law) or pressure-based solvers are activated to solve for compressible flows. The general NRBC is available for use in the **Pressure Inlet**, **Mass-Flow Inlet**, and **Velocity Inlet** dialog boxes when the pressure-based solver is activated to solve for compressible flows.

The example below shows you how to enable the general NRBC in the **Pressure Outlet** dialog box. You would follow similar steps for the other boundary condition types mentioned above.

1. Select **pressure-outlet** from the **Boundary Conditions** task page and click the **Edit...** button.

2. In the **Pressure Outlet** dialog box, choose **Non Reflecting** under **Acoustic Wave Model**.

Figure 6.68: The Pressure Outlet Dialog Box With the Non-Reflecting Boundary Enabled



3. For the density-based solver, select one of the two **Exit Pressure Specification** options: **Pressure at Infinity** or **Average Boundary Pressure**. The pressure-based solver uses the **Pressure at Infinity** option and the **Exit Pressure Specification** drop-down list is not available.

   a. The **Pressure at Infinity** boundary is typically used in unsteady calculations or when the exit pressure value is imposed at infinity. The boundary is designed so that the pressure at the boundary relaxes toward the imposed pressure at infinity. The speed at which this relaxation takes place is controlled by the parameter, `sigma`, which can be adjusted in the TUI:

   `define` → `boundary-conditions` → `non-reflecting-bc` → `general-nrbc` → `set`

   In the `set/` submenu, you can set the `sigma` value. The default value for `sigma` is 0.15.

   b. The **Average Boundary Pressure** specification is usually used in steady-state calculations when you want to force the average pressure on the boundary to approach the exit pressure value. The

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

matching of average exit pressure to the imposed average pressure is controlled by the parameter `sigma2` which can be adjusted in the TUI:

`define` → `boundary-conditions` → `non-reflecting-bc` → `general-nrbc` → `set`

In the `set/` submenu, you can set the `sigma2` value. The default value for `sigma2` is 5.0.

Important

There is no guarantee that the `sigma2` value of 5.0 will force the average boundary pressure to match the specified exit pressure in all flow situations. In the case where the desired average boundary pressure has not been achieved, you can intervene to adjust the `sigma2` value so that the desired average pressure on the boundary is approached.

4. For the pressure-based solver, you can select one of two **Backflow Pressure Specification** options: **Static Pressure** or **Total Pressure**. For the density-based solver, the **Backflow Pressure Specification** is not shown and the **Total Pressure** option is used by default. Note that backflow is not allowed for pressure inlets, mass-flow inlets, or velocity inlets.

   Important

   For the pressure-based solver, you should choose **Direction Vector** or **From Neighboring Cell** as the **Backflow Direction Specification Method** if the flow is tangential to the boundary. You should not select the **Normal to the Boundary** option for **Backflow Direction Specification Method** in this case because the face velocity components for this case will be computed from flux as zero during initialization. This initialization will cause the solver convergence to fail.

Usually, the solver can operate at higher CFL values without the NRBCs being turned on. Therefore, the best practice is to first achieve a good stable solution (not necessarily converged) before activating the non-reflecting boundary condition. In many flow situations, the CFL value must be reduced from the normal operation to keep the solution stable. This is particularly true with the density-based implicit solver since the boundary update is done in an explicit manner. A typical CFL value in the density-based implicit solver, with the NRBC activated, is 2.0 and 4.0 in the pressure-based solver.

6.4.3. Impedance Boundary Conditions

The impedance boundary condition (IBC) lies in between a traditional reflective boundary condition and a fully non-reflective boundary condition. It provides the ability to specify a partial reflection in the range from full-reflection to no-reflection. Impedance is a complex value; it is the reflection that changes the amplitude and the phase of the incoming wave. The use of impedance boundary conditions comes in cases where the flow in the simulation is highly influenced by reflected waves from objects outside the computational domain. In such cases the acoustic wave interaction from the larger domain can be modeled in the smaller domain through the use of impedance boundary conditions.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.4.3.1. Restrictions and Limitations

The impedance boundary condition is available only in the pressure-based solver. It is incompatible with steady-state flow, multiphase, or compressible liquid models (**compressible-liquid** method for density).

6.4.3.2. Theory

Impedance specifies an acoustic resistance in the frequency domain. It is a characteristic of the properties of the media and specific geometry described by a ratio of the pressure perturbation to the normal velocity perturbation at the boundary Blackstock [[11] (p. 2887)].

$$\hat{Z}(\omega)=\frac{\hat{p}'}{\rho c\hat{u}'} \tag{6.195}$$

where apostrophe denotes acoustic perturbation and hat denotes quantity in the frequency domain.

Fluent is a time domain solver. It cannot use the impedance from the frequency domain directly. The above expression and all its variables have to be converted to the time domain. After the conversion the relation between pressure and normal velocity perturbations is expressed through a convolution integral.

$$p'(t)=\int_{-\infty}^{\infty} Z(\tau)u'(t-\tau)d\tau \tag{6.196}$$

If impedance, $Z(\tau)$, is unbounded in the time domain, then admittance is used (admittance is the inverse of impedance). Fluent uses the reflection coefficient instead of impedance/admittance, to uniformly treat unbounded cases Fung [[33] (p. 2888)]. The reflection coefficient is a ratio between reflected and incoming wave amplitudes at the boundary. It is expressed through the impedance as:

$$\hat{R}(\omega)=\frac{\hat{Z}(\omega)-1}{\hat{Z}(\omega)+1} \tag{6.197}$$

Using a reflection coefficient, the relation between pressure and normal velocity perturbation is:

$$p'(t)-\rho cu'(t)=\int_{-\infty}^{\infty} R(\tau)\left[p'(t-\tau)+\rho cu'(t-\tau)\right]d\tau \tag{6.198}$$

The discretized form of this expression is used in Fluent to connect acoustic pressure and normal velocity. The computed acoustic perturbations are superimposed onto the pressure and velocity from non-reflecting boundary condition equations. The non-reflecting boundary condition equations provide mean flow values at the boundary, which drive the flow in the domain.

The data for the reflection coefficient are available in the frequency domain. As such they usually do not satisfy the causality and reality conditions. Fluent asks you to provide the reflection coefficient data in the form of a special approximation. This approximation is based on the system theory, which ensures that the reflection coefficient in the time domain will satisfy the above conditions Fung [[32] (p. 2888)]. The reflection coefficient is represented as a sum of zero, first and second order systems. The zero system is described with a real value, the first order system is described with a real pole, and the second order system is described with a pair of complex conjugate poles. Introducing a system variable, $s=j\omega$ the complete approximation for the reflection coefficient is:

$$\hat{R}(s)=D+\sum_{k=1}^{N_1}\frac{A_k}{s+\lambda_k}+\sum_{l=1}^{N_2}\left(\frac{B_l+jC_l}{s+\alpha_l-j\beta_l}+\frac{B_l-jC_l}{s+\alpha_l+j\beta_l}\right) \tag{6.199}$$

where D is a real term, N_1 is a number of real poles, λ_k and A_k are real pole and its amplitude, N_2 is a number of complex conjugate pole pairs, α_l, β_l are real and imaginary part of the complex conjugate pole, B_l and C_l are real and imaginary part of the amplitude of the complex pole.

To obey the causality and reality conditions, real pole λ_k, real α_l and imaginary part β_l of the complex conjugate pole should be positive. The passivity condition requires that the absolute value of zero order term D be less than 1. The above restrictions are enforced in the user interface. In addition you should ensure that the absolute value of the reflection coefficient computed by this formula is less than 1.

The impedance data can be obtained from measurements or from an acoustic solver. Running these data through a mathematics package will provide an approximation in terms of first and second order poles.

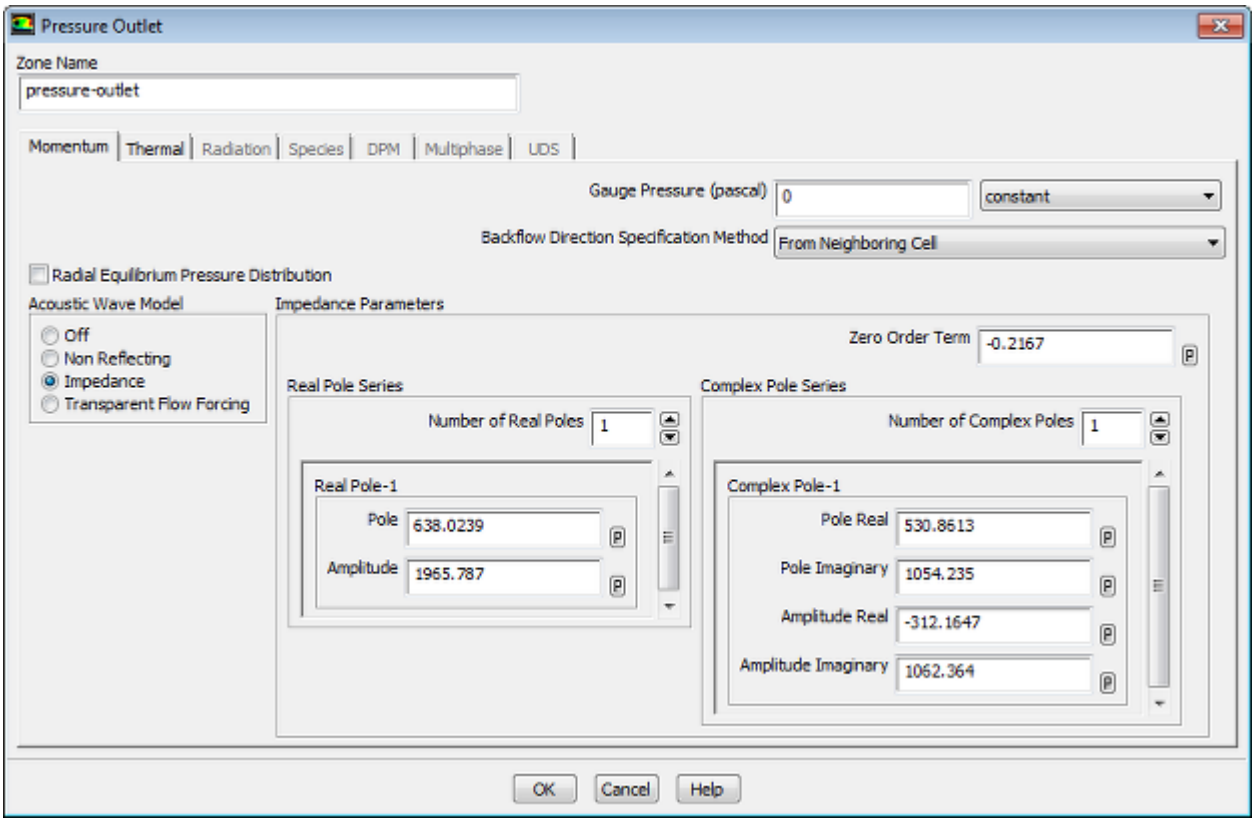
6.4.3.3. Using the Impedance Boundary Condition

The example below shows you how to enable the impedance boundary condition (IBC) in the **Pressure Outlet** dialog box. Similarly, you can enable IBC in the **Pressure Inlet**, **Velocity Inlet**, and **Mass-Flow Inlet** dialog boxes for compressible flows with the pressure-based solver.

1. Select **pressure-outlet** from the **Boundary Condition** task page and click the **Edit...** button.

2. In the **Pressure Outlet** dialog box, choose **Impedance** under **Acoustic Wave Model**.

   The dialog box will expand to reveal **Impedance Parameters**.



3. In **Impedance Parameters** enter data for the reflection coefficient according to the approximation formula (Equation 6.199 (p. 423)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Specify the **Zero Order Term**, D.

b. Under **Real Pole Series**, set the **Number of Real Poles** and for each real pole specify the **Pole**, λ_k, and the **Amplitude**, A_k.

c. Under **Complex Pole Series**, set the **Number of Complex Poles** and for each complex pole pair, specify **Pole Real**, α_l, **Pole Imaginary**, β_l, **Amplitude Real**, B_l, and **Amplitude Imaginary**, C_l.

Important

If flow is tangential to the boundary, then specify either **From Neighboring Cell** or **Direction Vector** for **Back flow Direction Specification Method**. Do not select **Normal to the boundary** because this computes the face velocity components from flux to zero values during initialization, which impairs solver convergence.

The IBC is implemented on top of the non-reflective boundary condition. Choose a time step that will not make the CFL number exceed a value of 1 in the cells adjacent to the impedance boundary.

Note

The mean flow in the domain should be well established before enabling IBC.

6.4.4. Transparent Flow Forcing Boundary Conditions

It is often desirable in simulations to cut a domain down to a smaller region to model only a part of the larger domain. The treatment at the boundaries created when doing so should be consistent with a slice through the larger domain. That is, incoming transients should be allowed to enter the computational domain and outgoing transients should leave the domain without reflections. Artificial numerical reflections at the boundaries can significantly affect the predicted flow, particularly if the geometry of the numerical domain has Eigen-frequencies that match the frequencies of physical waves propagating through the flow.

The transparent flow forcing treatment provides the capability to model incoming waves while allowing outgoing waves to pass through the boundaries without reflection. It is available in transient simulations of compressible flow using the pressure-based solver and can be applied for velocity inlets, mass-flow inlets, pressure inlets, and pressure outlets.

6.4.4.1. Restrictions and Limitations

The transparent flow forcing condition is available only in the pressure-based solver and it is incompatible with steady-state flow, multiphase, and compressible liquid models (those using the compressible-liquid method for density).

6.4.4.2. Theory

The transparent flow forcing boundary condition is implemented on top of the non-reflecting boundary condition. The non-reflecting part of the boundary condition drives the mean flow. The transient flow at the boundary is split into two waves [50] (p. 2889): incoming wave, $W_{in}\left(\vec{r},t\right)$, and outgoing wave,

$W_{out}(\vec{r},t)$. In the acoustic limit, the intensities of the waves are expressed through pressure and normal velocity perturbations as:

$$W_{in}(\vec{r},t)=p'+zu'_n$$
$$W_{out}(\vec{r},t)=p'-zu'_n$$

where $z=\rho c$ is the specific impedance of the mean flow with density, ρ, and speed of sound, c. The outgoing wave intensity, $W_{out}(\vec{r},t)$, is computed internally while the incoming wave intensity, $W_{in}(\vec{r},t)$, is user-specified through a user-defined profile.

6.4.4.3. Using the Transparent Flow Forcing Boundary Condition

The transparent flow forcing (TFF) boundary condition is available at velocity inlets, mass-flow inlets, pressure inlets, and pressure outlets when simulating transient compressible flow using the pressure-based solver. The steps for using the transparent flow forcing boundary condition are as follows:

1. Create a profile for the incoming wave intensity on the boundary. The profile can be created either as a profile file as described in Profiles (p. 435) or using the `DEFINE_PROFILE` macro as described in Fluent Customization Manual. In the profile definition, you will specify the wave intensity as a function of position and time. For the incoming wave, the pressure and normal velocity perturbations satisfy the relation:

   $$p'=zu'_n$$

   Using this relation the incoming wave intensity at the boundary can be computed from any of the flow perturbation quantities.

   - from acoustic pressure, p'

     $$W_{in}=2p'$$

   - from acoustic velocity, u'_n

     $$W_{in}=2zu'_n$$

   - from acoustic mass flux, m'_n

     $$W_{in}=2cm'_n$$

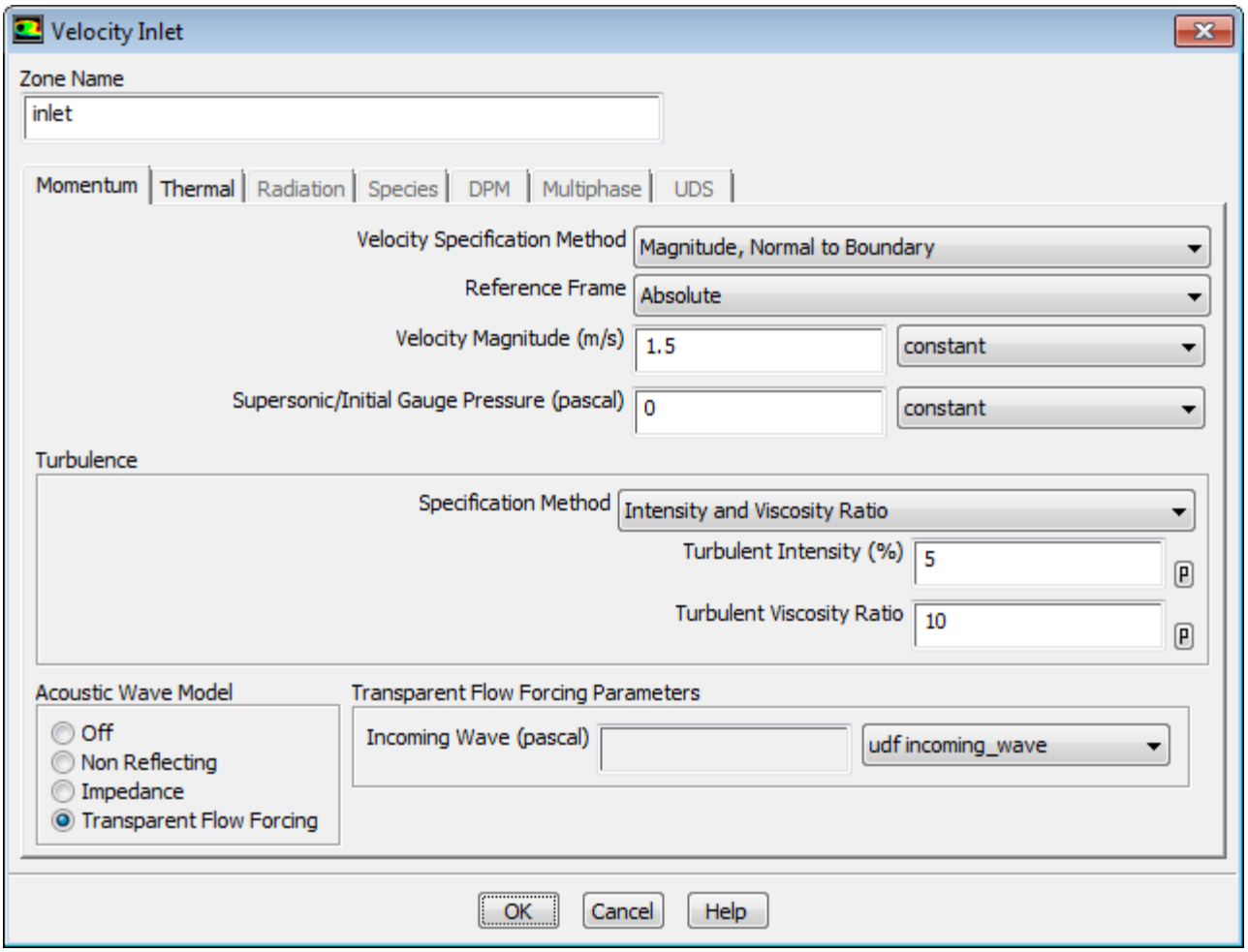
   where c is the speed of sound and $z=\rho c$ is the acoustic impedance.

2. Open the boundary condition settings dialog box for the boundary zone of interest (Setting Cell Zone and Boundary Conditions (p. 241)).

3. On the **Momentum** tab, select **Transparent Flow Forcing** under **Acoustic Wave Model**.

4. Under **Transparent Flow Forcing Parameters**, select your profile from the drop-down list next to **Incoming Wave**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Important

- If flow is tangential to the boundary, then specify either **From Neighboring Cell** or **Direction Vector** for **Back flow Direction Specification Method**. Do not select **Normal to the boundary** because this computes the face velocity components from flux to zero values during initialization, which impairs solver convergence.
- The TFF condition is implemented on top of the non-reflective boundary condition. Choose a time step that will not make the CFL number exceed a value of 1 in the cells adjacent to the TFF boundary.
- The mean flow in the domain should be well established before enabling TFF.

6.5. User-Defined Fan Model

The user-defined fan model in ANSYS Fluent allows you to periodically regenerate a profile file that can be used to specify the characteristics of a fan, including pressure jump across the fan, and radial and swirling components of velocity generated by the fan.

For example, consider the calculation of the pressure jump across the fan. You can, through the standard interface, input a constant for the pressure jump, specify a polynomial that describes the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

pressure jump as a function of axial velocity through the fan, or use a profile file that describes the pressure jump as a function of the axial velocity or location at the fan face. If you use a profile file, the same profile will be used consistently throughout the course of the solution. Suppose, however, that you want to change the profile as the flow field develops. This would require a periodic update to the profile file itself, based upon some instructions that you supply. The user-defined fan model is designed to help you do this.

To use this model, you need to generate an executable that reads a fan profile file that is written by ANSYS Fluent, and writes out a modified one, which ANSYS Fluent will then read. The source code for this executable can be written in any programming language (Fortran or C, for example). Your program will be called and executed automatically, according to inputs that you supply through the standard interface.

Information about the user-defined fan model is provided in the following sections.

6.5.1. Steps for Using the User-Defined Fan Model
6.5.2. Example of a User-Defined Fan

6.5.1. Steps for Using the User-Defined Fan Model

To make use of the user-defined fan model, follow the steps below.

1. In your model, identify one or more interior faces to represent one or more fan zones.

   Setup → **Boundary Conditions**

2. Input the name of your executable and the instructions for reading and writing profile files in the User-Defined Fan Model Dialog Box (p. 2824).

   User Defined → **Model Specific** → **Fan Model...**

3. Initialize the flow field and the profile files.

4. Enter the fan parameters using the standard Fan Dialog Box (p. 2397) (opened from the Boundary Conditions Task Page (p. 2389)).

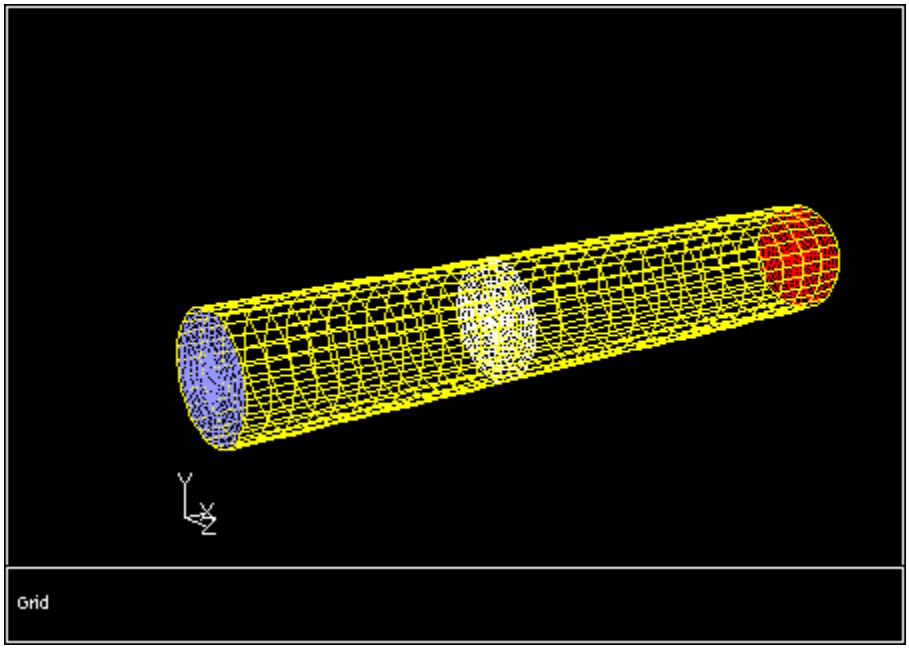
5. Perform the calculation.

6.5.2. Example of a User-Defined Fan

Usage of the user-defined fan model is best demonstrated by an example. With this in mind, consider the domain shown in Figure 6.69: The Inlet, Fan, and Pressure Outlet Zones for a Circular Fan Operating in a Cylindrical Domain (p. 429). An inlet supplies air at 10 m/s to a cylindrical region, 1.25 m long and 0.2 m in diameter, surrounded by a symmetry boundary. At the center of the flow domain is a circular fan. A pressure outlet boundary is at the downstream end.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.69: The Inlet, Fan, and Pressure Outlet Zones for a Circular Fan Operating in a Cylindrical Domain



Solving this problem with the user-defined fan model will cause ANSYS Fluent to periodically write out a radial profile file with the current solution variables at the fan face. These variables (static pressure, pressure jump, axial, radial, and swirling (tangential) velocity components) will represent averaged quantities over annular sections of the fan. The sizes of the annular regions are determined by the size of the fan and the number of radial points to be used in the profiles.

Once the profile file is written, ANSYS Fluent will invoke an executable, which will perform the following tasks:

1. Read the profile file containing the current flow conditions at the fan.
2. Perform a calculation to compute new values for the pressure jump, radial velocity, and swirl velocity for the fan.
3. Write a new profile file that contains the results of these calculations.

ANSYS Fluent will then read the new profile file and continue with the calculation.

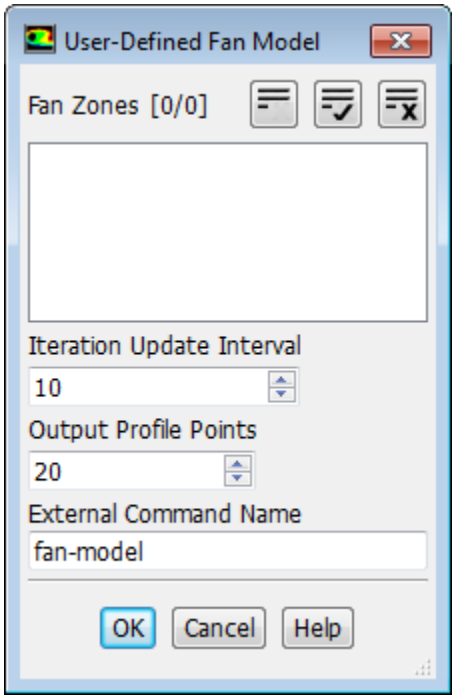
6.5.2.1. Setting the User-Defined Fan Parameters

Specification of the parameters for the user-defined fan begins in the User-Defined Fan Model Dialog Box (p. 2824) (Figure 6.70: The User-Defined Fan Model Dialog Box (p. 430)).

User Defined → Model Specific → Fan Model...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.70: The User-Defined Fan Model Dialog Box



In this dialog box, you can select the fan zone(s) on which your executable will operate under **Fan Zones**. In this example, there is only one fan, **fan-8**. If you have multiple fan zones in a simulation, for which you have different profile specifications, you can select them all at this point. Your executable will be able to differentiate between the fan zones because the zone ID for each fan is included in the solution profile file. The executable will be invoked once for each zone, and separate profile files will be written for each.

The executable file will be called on to update the profile file periodically, based on the input for the **Iteration Update Interval**. An input of `10`, as shown in the dialog box, means that the fan executable in this example will act every 10 iterations to modify the profile file.

The number of points in the profile file to be written by ANSYS Fluent is entered under **Output Profile Points**. This profile file can have the same or a different number of points as the one that is written by the external executable.

Finally, the name of the executable should be entered under **External Command Name**. In the current example, the name of the executable is `fan-model`.

Important

If the executable is not located in your working directory, then you must type the complete path to the executable.

6.5.2.2. Sample User-Defined Fan Program

The executable file will be built from the Fortran program, `fantest.f`, which is shown below. You can obtain a copy of this subroutine and the two that it calls (to read and write profile files) by contacting your ANSYS Fluent technical support engineer.

```
c
c  This program is invoked at intervals by ANSYS Fluent to
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c  read a profile-format file that contains radially
c  averaged data at a fan face, compute new pressure-jump
c  and swirl-velocity components, and write a new profile
c  file that will subsequently be read by ANSYS Fluent to
c  update the fan conditions.
c
c  Usage: fantest input_profile output_profile
c

  integer npmax
  parameter (npmax = 900)
  integer inp         ! input: number of profile points
  integer iptype      ! input: profile type (0=radial, 1=point)
  real ir(npmax)      ! input: radial positions
  real ip(npmax)      ! input: pressure
  real idp(npmax)     ! input: pressure-jump
  real iva(npmax)     ! input: axial velocity
  real ivr(npmax)     ! input: radial velocity
  real ivt(npmax)     ! input: tangential velocity
  character*80 zoneid
  integer rfanprof    ! function to read a profile file
  integer status
c
  status = rfanprof(npmax,zoneid,iptype,
 $  inp,ir,ip,idp,iva,ivr,ivt)
  if (status.ne.0) then
    write(*,*) 'error reading input profile file'
  else
    do 10 i = 1, inp
      idp(i) = 200.0 - 10.0*iva(i)
      ivt(i) = 20.0*ir(i)
      ivr(i) = 0.0
 10  continue
  call wfanprof(6,zoneid,iptype,inp,ir,idp,ivr,ivt)
  endif
  stop
  end
```

After the variable declarations, which have comments on the right, the subroutine `rfanprof` is called to read the profile file, and pass the current values of the relevant variables (as defined in the declaration list) to `fantest`. A loop is done on the number of points in the profile to compute new values for:

- The pressure jump across the fan, `idp`, which in this example is a function of the axial velocity, `iva`.
- The swirling or tangential velocity, `ivt`, which in this example is proportional to the radial position, `ir`.
- The radial velocity, `ivr`, which in this example is set to zero.

After the loop, a new profile is written by the subroutine `wfanprof`, shown below. (For more information on profile file formats, see Profile File Formats (p. 436).)

```
 subroutine wfanprof(unit,zoneid,ptype,n,r,dp,vr,vt)
c
c  writes an ANSYS Fluent profile file for input by the
c  user fan model
c
  integer unit       ! output unit number
  character*80 zoneid
  integer ptype      ! profile type (0=radial, 1=point)
  integer n          ! number of points
  real r(n)          ! radial position
  real dp(n)         ! pressure jump
  real vr(n)         ! radial velocity
  real vt(n)         ! tangential velocity
  character*6 typenam
  if (ptype.eq.0) then
    typenam = 'radial'
  else
    typenam = 'point'
```

```
 endif
 write(unit,*) '((', zoneid(1:index(zoneid,'\0')-1), ' ',
$  typenam, n, ')'

 write(unit,*) '(r'
 write(unit,100) r
 write(unit,*) ')'

 write(unit,*) '(pressure-jump'
 write(unit,100) dp
 write(unit,*) ')'

 write(unit,*) '(radial-velocity'
 write(unit,100) vr
 write(unit,*) ')'

 write(unit,*) '(tangential-velocity'
 write(unit,100) vt
 write(unit,*) ')'

100 format(5(e15.8,1x))
 return
 end
```

This subroutine will write a profile file in either radial or point format, based on your input for the integer `ptype`. (See Profiles (p. 435) for more details on the types of profile files that are available.) The names that you use for the various profiles are arbitrary. Once you have initialized the profile files, the names you use in `wfanprof` will appear as profile names in the Fan Dialog Box (p. 2397).

6.5.2.3. Initializing the Flow Field and Profile Files

The next step in the setup of the user-defined fan is to initialize (create) the profile files that will be used. To do this, first initialize the flow field with the Solution Initialization Task Page (p. 2520) (using the velocity inlet conditions, for example), and then type the command `(update-user-fans)` in the console window. (The parentheses are part of the command, and must be typed in.)

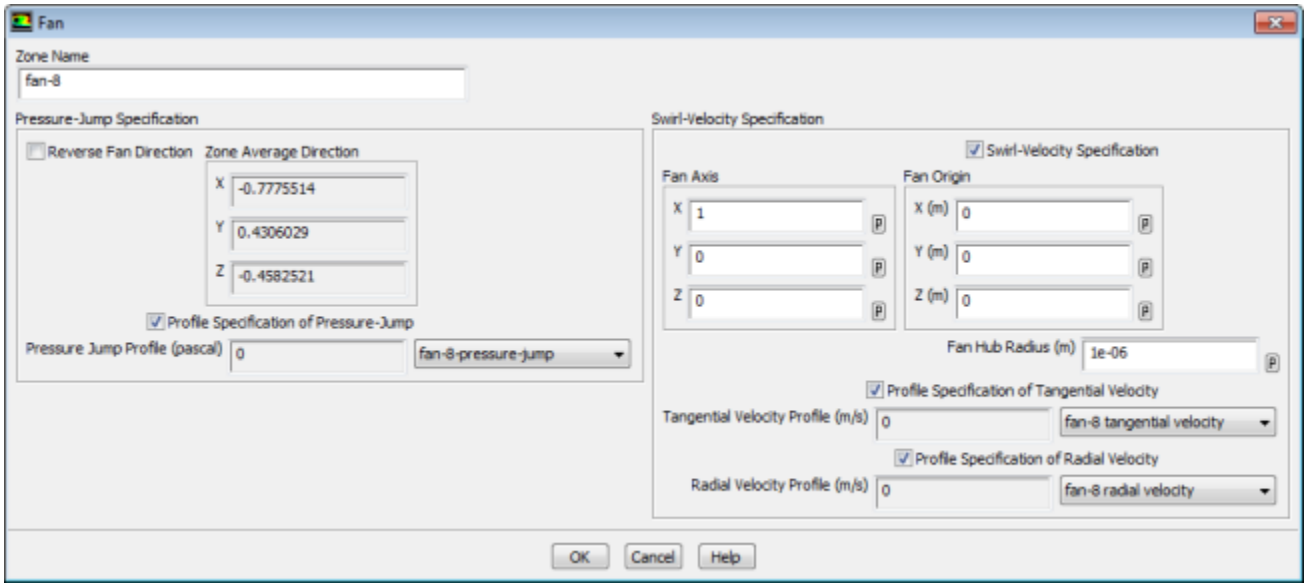
This will create the profile names that are given in the subroutine `wfanprof`.

6.5.2.4. Selecting the Profiles

Once the profile names have been established, you will need to visit the Fan Dialog Box (p. 2397) (Figure 6.71: The Fan Dialog Box (p. 433)) to complete the problem setup. (See Fan Boundary Conditions (p. 389) for general information on using the **Fan** dialog box.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.71: The Fan Dialog Box



At this time, the **Fan Axis**, **Fan Origin**, and **Fan Hub Radius** can be entered, along with the choice of profiles for the calculation of pressure jump, tangential velocity, and radial velocity. With the profile options enabled, you can select the names of the profiles from the drop-down lists. In the dialog box above, the selected profiles are named **fan-8-pressure-jump**, **fan-8 tangential velocity**, and **fan-8 radial velocity**, corresponding to the names that were used in the subroutine `wfanprof`.

6.5.2.5. Performing the Calculation

The solution is now ready to run. As it begins to converge, the report in the console window shows that the profile files are being written and read every 10 iterations:

```
    iter   continuity   x-velocity   y-velocity   z-velocity          k
!      1   residual normalization factors changed (continuity
       1   1.0000e+00   1.0000e+00   1.0000e+00   1.0000e+00   1.0000e+00
!      2   residual normalization factors changed (continuity
       2   1.0000e+00   1.0000e+00   1.0000e+00   1.0000e+00   9.4933e-01
       3   6.8870e-01   7.2663e-01   7.3802e-01   7.5822e-01   6.1033e-01
       .       .            .            .            .            .
       .       .            .            .            .            .
       .       .            .            .            .            .
       9   2.1779e-01   9.8139e-02   3.0497e-01   2.9609e-01   2.8612e-01
 Writing "fan-8-out.prof"...
 Done.
 Reading "fan-8-in.prof"...

 Reading profile file...
      10   "fan-8"  radial-profile points, r, pressure-jump,
                          radial-velocity, tangential-velocity.

 Done.
      10   1.7612e-01   7.4618e-02   2.5194e-01   2.4538e-01   2.4569e-01
      11   1.6895e-01   8.3699e-02   2.0316e-01   2.0280e-01   2.1169e-01
       .       .            .            .            .            .
       .       .            .            .            .            .
       .       .            .            .            .            .
```

The file `fan-8-out.prof` is written out by ANSYS Fluent and read by the executable `fantest`. It contains values for pressure, pressure jump, axial velocity, radial velocity, and tangential velocity at 20 radial locations at the site of the fan. The file `fan-8-in.prof` is generated by `fantest` and contains updated values for pressure jump and radial and tangential velocity only. It is therefore a smaller file

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

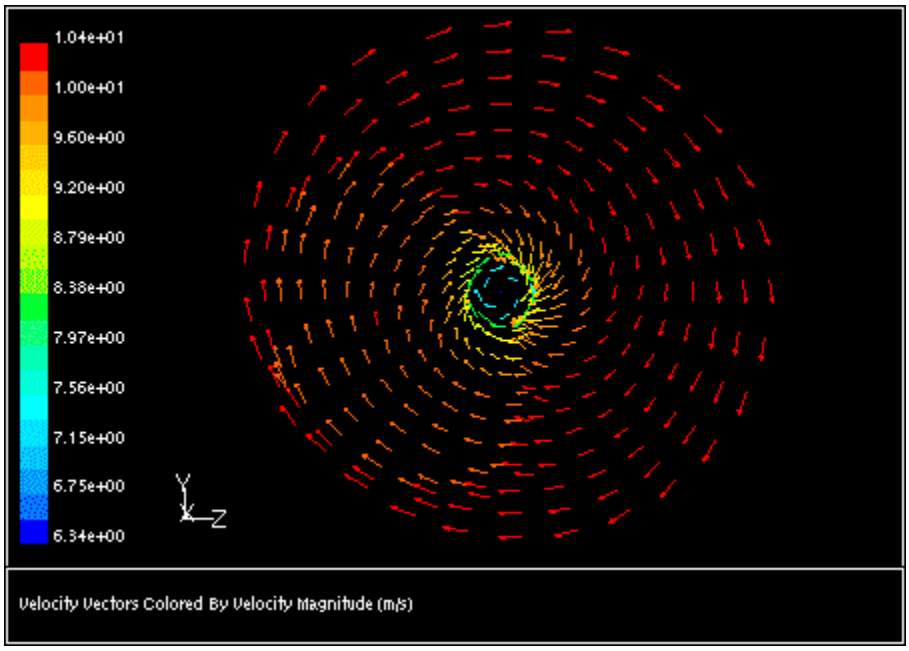
than `fan-8-out.prof`. The prefix for these files takes its name from the fan zone with which the profiles are associated. An example of the profile file `fan-8-in.prof` is shown below. This represents the last profile file to be written by `fantest` during the convergence history.

```
((fan-8 radial 10)
(r
0.24295786E-01 0.33130988E-01 0.41966137E-01 0.50801374E-01 0.59636571E-01
0.68471842E-01 0.77307090E-01 0.86142287E-01 0.94963484E-01 0.95353782E-01
)
(pressure-jump
0.10182057E+03 0.98394081E+02 0.97748657E+02 0.97787750E+02 0.97905228E+02
0.98020668E+02 0.98138817E+02 0.98264198E+02 0.98469681E+02 0.98478783E+02
)
(radial-velocity
0.00000000E+00 0.00000000E+00 0.00000000E+00 0.00000000E+00 0.00000000E+00
0.00000000E+00 0.00000000E+00 0.00000000E+00 0.00000000E+00 0.00000000E+00
)
(tangential-velocity
0.48591572E+00 0.66261977E+00 0.83932275E+00 0.10160275E+01 0.11927314E+01
0.13694369E+01 0.15461419E+01 0.17228458E+01 0.18992697E+01 0.19070756E+01
)
```

6.5.2.6. Results

A plot of the transverse velocity components at the site of the fan is shown in Figure 6.72: Transverse Velocities at the Site of the Fan (p. 434). As expected, there is no radial component, and the tangential (swirling) component increases with radius.

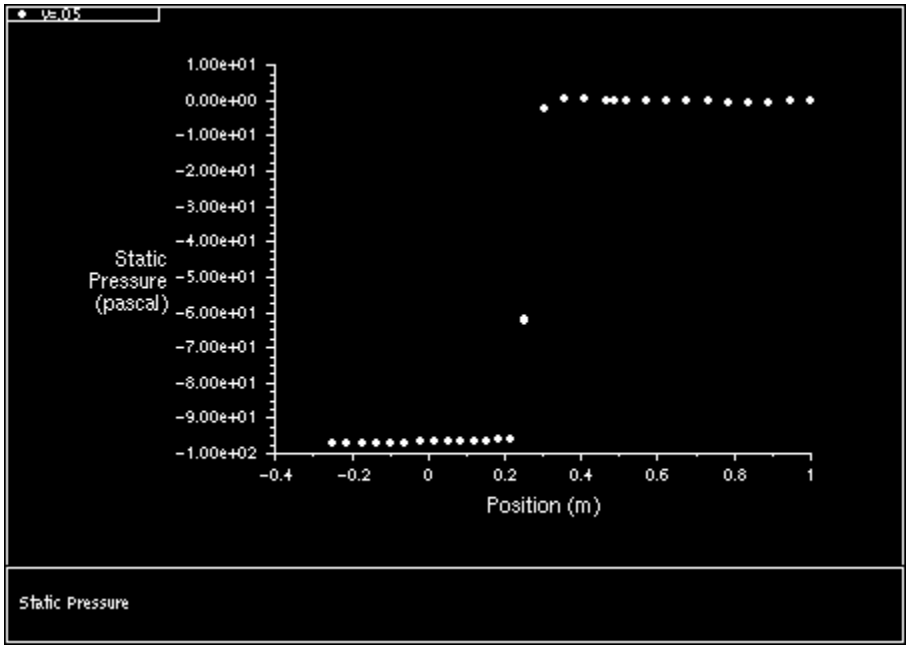
Figure 6.72: Transverse Velocities at the Site of the Fan



As a final check on the result, an XY plot of the static pressure as a function of x position is shown (Figure 6.73: Static Pressure Jump Across the Fan (p. 435)). This XY plot is made on a line at y=0.05 m, or at about half the radius of the duct. According to the input file shown above, the pressure jump at the site of the fan should be approximately 97.8 Pa/m. Examination of the figure supports this finding.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.73: Static Pressure Jump Across the Fan



6.6. Profiles

Profiles can be boundary conditions, cell zone conditions, and initial conditions for discrete phases. ANSYS Fluent provides a very flexible profile definition mechanism. This feature allows you to use experimental data, data calculated by an external program, or data written from a previous solution using the Write Profile Dialog Box (p. 2388) (as described in Reading and Writing Profile Files (p. 53)) as the boundary condition for a variable.

Information about profiles is presented in the following subsections:

6.6.1. Profile Specification Types
6.6.2. Profile File Formats
6.6.3. Using Profiles
6.6.4. Reorienting Profiles
6.6.5. Defining Transient Cell Zone and Boundary Conditions

6.6.1. Profile Specification Types

The following is a list of the six types of profiles that can be read into ANSYS Fluent, as well as information about the interpolation method employed by ANSYS Fluent for each type.

- Point profiles are specified by an unordered set of n points: $\left(x_i, y_i, v_i\right)$ for 2D problems or $\left(x_i, y_i, z_i, v_i\right)$ for 3D problems, where $1 \le i \le n$. Profiles written using the **Write Profile** dialog box and profiles of experimental data in random order are examples of point profiles.

  ANSYS Fluent will interpolate the point cloud to obtain values at the boundary faces. The default interpolation method for the unstructured point data is zeroth order. That is, for each cell face at the boundary, the solver uses the value from the profile file located closest to the cell. Therefore, to get an accurate specification of an inlet profile using the default interpolation method, your profile file should contain a sufficiently high point density. For information about other available interpolation methods for point profiles, see Using Profiles (p. 440).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Line profiles are specified for 2D problems by an ordered set of n points: $\left(x_i, y_i, v_i\right)$, where $1 \le i \le n$. Zeroth-order interpolation is performed between the points. An example of a line profile is a profile of data obtained from an external program that calculates a boundary-layer profile.

- Mesh profiles are specified for 3D problems by an m by n mesh of points: $\left(x_{ij}, y_{ij}, z_{ij}, v_{ij}\right)$, where $1 \le i \le m$ and $1 \le j \le n$. Zeroth-order interpolation is performed between the points. Examples of mesh profiles are profiles of data from a structured mesh solution and experimental data in a regular array.

- Radial profiles are specified for 2D and 3D problems by an ordered set of n points: $\left(r_i, v_i\right)$, where $1 \le i \le n$. The data in a radial profile are a function of radius only. Linear interpolation is performed between the points, which must be sorted in ascending order of the r field. The axis for the cylindrical coordinate system is determined as follows:

  – For 2D problems, it is the z-direction vector through (0,0).

  – For 2D axisymmetric problems, it is the x-direction vector through (0,0).

  – For 3D problems involving a swirling fan, it is the fan axis defined in the Fan Dialog Box (p. 2397) (unless you are using local cylindrical coordinates at the boundary, as described below).

  – For 3D problems without a swirling fan, it is the rotation axis of the adjacent fluid zone, as defined in the Fluid Dialog Box (p. 2369) (unless you are using local cylindrical coordinates at the boundary, as described below).

  – For 3D problems in which you are using local cylindrical coordinates to specify conditions at the boundary, it is the axis of the specified local coordinate system.

- Axial profiles are specified for 3D problems by an ordered set of n points: $\left(z_i, v_i\right)$, where $1 \le i \le n$. The data in an axial profile are a function of the axial direction. Linear interpolation is performed between the points, which must be sorted in ascending order of the z field.

- Transient profiles are specified for 2D and 3D profiles by an ordered set of n points: $\left(t_i, v_{0,i}, v_{1,i}, v_{2,i}, \ldots\right)$. Linear interpolations are done between the points which must be sorted in ascending order of the t (time or crank angle) field. Examples of transient profiles are transient cell zone and boundary conditions (see Defining Transient Cell Zone and Boundary Conditions (p. 448)) and point properties for particle injections (see Point Properties for Transient Injections (p. 1300)).

6.6.2. Profile File Formats

There are two profile format options:
6.6.2.1. Standard Profiles
6.6.2.2. CSV Profiles

6.6.2.1. Standard Profiles

The format of the profile files is fairly simple. The file can contain an arbitrary number of profiles. Each profile consists of a header that specifies the profile name, profile type (`point`, `line`, `mesh`, `radial`,

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

or axial), and number of defining points, and is followed by an arbitrary number of named "fields". Some of these fields contain the coordinate points and the rest contain boundary data.

Important

Keep profile names to 64 characters or less. If the profile name contains more than 64 characters, Fluent truncates the name to the first 64 characters.

All quantities, including coordinate values, must be specified in SI units because ANSYS Fluent does not perform unit conversion when reading profile files.

Parentheses are used to delimit profiles and the fields within the profiles. Any combination of tabs, spaces, and newlines can be used to separate elements.

Important

In the general format description below, " | " indicates that you should input only one of the items separated by |'s and " ... " indicates a continuation of the list.

```
((profile1-name point|line|radial n)
 (field1-name a1 a2 ... an)
 (field2-name b1 b2 ... bn)
.
.
.
 (fieldf-name f1 f2 ... fn))

((profile2-name mesh m n)
  (field1-name  a11 a12 ... a1n
                a21 a22 ... a2n
                .
                .
                .
                am1 am2 ... amn)
.
.
.
(fieldf-name   f11  f12 ... f1n
               f21  f22 ... f2n
               .
               .
               .
               fm1 fm2 ... fmn))
```

Profile names must have all lowercase letters (for example, name). Uppercase letters in profile names are not acceptable. Each profile of type point, line, and mesh must contain fields with names x, y, and, for 3D, z. Each profile of type radial must contain a field with name r. Each profile of type axial must contain a field with name z. The rest of the names are arbitrary, but must be valid Scheme symbols. For compatibility with old-style profile files, if the profile type is missing, point is assumed.

6.6.2.1.1. Example

A typical usage of a profile file is to specify the profile of the boundary layer at an inlet. For a compressible flow calculation, this will be done using profiles of total pressure, k, and ε. For an incompressible flow, it might be preferable to specify the inlet value of streamwise velocity, together with k and ε.

Below is an example of a profile file that does this:

```
((turb-prof point 8)
 (x
     4.00000E+00  4.00000E+00  4.00000E+00  4.00000E+00
     4.00000E+00  4.00000E+00  4.00000E+00  4.00000E+00 )
 (y
     1.06443E-03  3.19485E-03  5.33020E-03  7.47418E-03
     2.90494E-01  3.31222E-01  3.84519E-01  4.57471E-01 )
 (u
     5.47866E+00  6.59870E+00  7.05731E+00  7.40079E+00
     1.01674E+01  1.01656E+01  1.01637E+01  1.01616E+01 )
 (tke
     4.93228E-01  6.19247E-01  5.32680E-01  4.93642E-01
     6.89414E-03  6.89666E-03  6.90015E-03  6.90478E-03 )
 (eps
     1.27713E+02  6.04399E+01  3.31187E+01  2.21535E+01
     9.78365E-03  9.79056E-03  9.80001E-03  9.81265E-03 )
 )
```

6.6.2.2. CSV Profiles

CSV profiles are compatible with spreadsheets and are available for both reading and writing in Fluent. Many profile types can be written in CSV format, including:

- Radial
- Axial
- Point (X, Y, Z)
- Transient
- Transient-Periodic

If writing a profile using the **Write Profile** dialog box, you can save the profile in CSV format by saving with a `.csv` extension.

Formatting Rules:

CSV profiles must be setup so they can be read and interpreted correctly in Fluent:

1. The profile file must contain the section identifiers shown in Table 6.4: **CSV Profile Section Identifiers** (p. 438). These identifiers must be used in the order, `[Name]` then `[Data]`.

Table 6.4: CSV Profile Section Identifiers

Section Identifiers	Remark
[Name]	Next line should have the profile name. *Note- if the profile name has space characters, they will be replaced with '_' (space character is not allowed).*
[Data]	Data values begin on the next line.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. After the [Data] identifier you must label the type of data that you are providing; refer to Table 6.5: Profile Types and the Corresponding Required Field Labels (p. 439) for the correct labels. If Fluent does not find one of the expected labels, the profile read will fail.

Table 6.5: Profile Types and the Corresponding Required Field Labels

Profile Type	Required Field Label
radial	radius
axial	axis
point	X, Y, Z
transient	time, angle
transient-periodic	time-periodic, angle-periodic

3. If there is an unsupported section identifier, Fluent ignores the section and looks for the [Name] and [Data] section identifiers.

4. Multiple profiles may be included in a single file.

Important

- Only use commas as value separators. Space characters should not be used as they will confuse external spreadsheet applications.
- Only SI values are supported. Quantities cannot be entered with units.

CSV Profile File Examples

A file containing one profile:

```
[Name]
outlet

[Data]
x,y,z,x-velocity
-0.00036448459,0.0068932362,3,0.5
0.0014999653,-0.0090896944,3,0.5
-0.0014358073,-0.0094413245,3,    0.5
0.0022810854,0.014916174,3,0.5
-0.0024638004,0.014873175,3,0.5
-0.0069573424,0.013364096,3,0.5
```

A file containing two profiles, the first one is spatial (steady-state) and the second is transient.

```
[Name]
outlet

[Data]
x,y,z,x-velocity
-0.00036448459,0.0068932362,3,0.05
0.0014999653,-0.0090896944,3,0.05


[Name]
transient-temperature
[Data]
time,temperature
1.1,300
```

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
1.2,350
1.3,400
```

6.6.3. Using Profiles

The procedure for using a profile to define a particular cell zone or boundary condition is outlined below.

1. Create a file that contains the desired profile, following the format described in Profile File Formats (p. 436).

2. Read the profile using the **Read...** button in the Profiles Dialog Box (p. 2385) (Figure 6.74: The Profiles Dialog Box (p. 442))

   or the **File/Read/Profile...** ribbon tab item.

   Setup → **Cell Zone Conditions** → **Profiles...**

   Setup → **Boundary Conditions** → **Profiles...**

   File → **Read** → **Profile...**

   Note that if you use the **Profiles** dialog box to read a file, and a profile in the file has the same name as an existing profile, the old profile will be overwritten.

3. If it is a point profile, you can choose the method of interpolation using the **Profiles** dialog box (Figure 6.74: The Profiles Dialog Box (p. 442)):

   Setup → **Cell Zone Conditions** → **Profiles...**

   Setup → **Boundary Conditions** → **Profiles...**

   Select the point profile in the **Profile** selection list. Then select one of the three choices in the **Interpolation Method** list and click the **Apply** button. The three choices include:

   - **Constant**

     This method is zeroth-order interpolation. For each cell face at the boundary, the solver uses the value from the profile file located closest to the cell. Therefore, the accuracy of the interpolated profile will be affected by the density of the data points in your profile file. This is the default interpolation method for point profiles.

   - **Inverse Distance**

     This method assigns a value to each cell face at the boundary based on weighted contributions from the values in the profile file. The weighting factor is inversely proportional to the distance between the profile point and the cell face center.

   - **Least Squares**

     This method assigns values to the cell faces at the boundary through a first-order interpolation method that tries to minimizes the sum of the squares of the offsets (residuals) between the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

profile data points and the cell face centers. The least squares solution is found using Singular Value Decomposition (SVD).

Note

The **Inverse Distance** and **Least Squares** profile interpolation methods are not applicable when a profile is attached to cell zones.

For information about the interpolation methods employed for other profile types (that is, line, mesh, radial, or axial profiles), see Profile Specification Types (p. 435).

4. In the boundary conditions dialog boxes (for example, the **Velocity Inlet** and **Pressure Inlet** dialog boxes), the fields defined in the profile file (and those defined in any other profile file that you have read in) will appear in the drop-down list to the right of or below each parameter for which profile specification is allowed. To use a particular profile, select it in the appropriate list.

5. Initialize the solution to interpolate the profile.

Note

You can use a profile file or the `DEFINE_PROFILE` user-defined function to specify volumetric source terms. If you specify the source terms with a profile, you will not have access to the central coefficient of the equations solved in order to linearize the source term. You will need to use a user-defined function to do this.

For more information on UDFs, refer to the Fluent Customization Manual.

6.6.3.1. Checking and Deleting Profiles

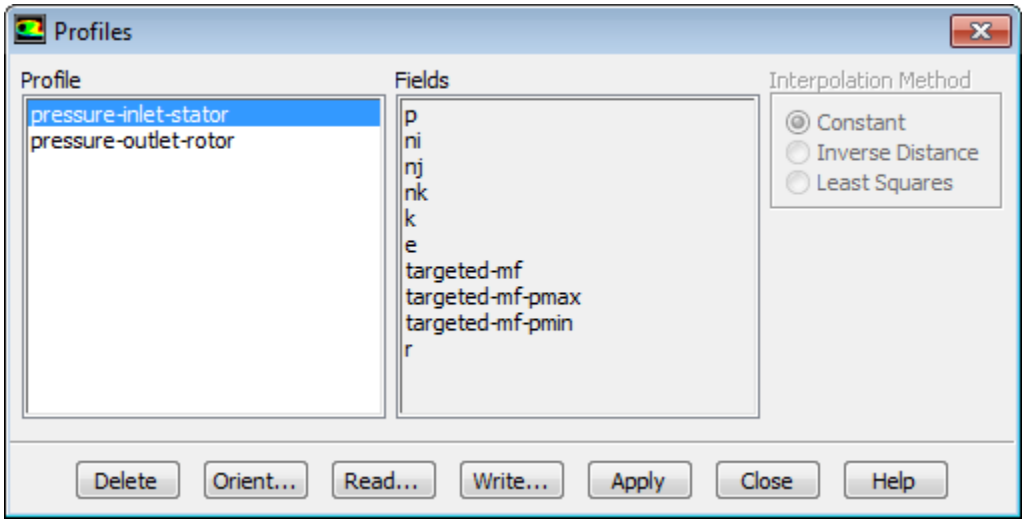
Each profile file contains one or more profiles, and each profile has one or more fields defined in it. Once you have read in a profile file, you can check which fields are defined in each profile, and you can also delete a particular profile. These tasks are accomplished in the Profiles Dialog Box (p. 2385) (Figure 6.74: The Profiles Dialog Box (p. 442)).

Setup → **Cell Zone Conditions** → **Profiles...**

Setup → **Boundary Conditions** → **Profiles...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.74: The Profiles Dialog Box



To check which fields are defined in a particular profile, select the profile name in the **Profile** list. The available fields in that file will be displayed in the **Fields** list. In Figure 6.74: The Profiles Dialog Box (p. 442), the profile fields from the profile file of Example (p. 437) are shown.

To delete a profile, select it in the **Profile** list and click the **Delete** button. When a profile is deleted, all fields defined in it will be removed from the **Fields** list.

Important

If you are deleting an existing profile to replace it with a new profile, use the following procedure:

1. Delete the existing profile.
2. Change the parameters defined by the existing profile to **Constant**.
3. Read the new profile and apply as desired.

6.6.3.2. Viewing Profile Data

The **Plots** task page options allow you to generate XY plots of data related to profiles. You can plot the original data points from the profile file you have read into ANSYS Fluent, or you can plot the values assigned to the cell faces on the boundary after the profile file has been interpolated. See XY Plots of Profiles (p. 1846) for the steps to generate these plots.

You have the additional option of viewing the parameters) using the **Plot** or the **Contours** options. Note that these display options do not allow you to plot the actual values of the cell faces (as is done with the **Interpolated Data** option), because they interpolate the values stored in the adjacent cells. To view the boundary condition parameters you must first read in the profile, save a boundary condition with a profile field selected as a parameter, and initialize the flow solution. Then you can view the surface data as follows:

- For 2D calculations, open the **Solution XY Plot** dialog box. Select the appropriate boundary zone in the **Surfaces** list, the variable of interest in the **Y Axis Function** drop-down list, and the desired **Plot Direction**. Ensure that the **Node Values** check button is turned on, and then click **Plot**. You should then see the profile

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

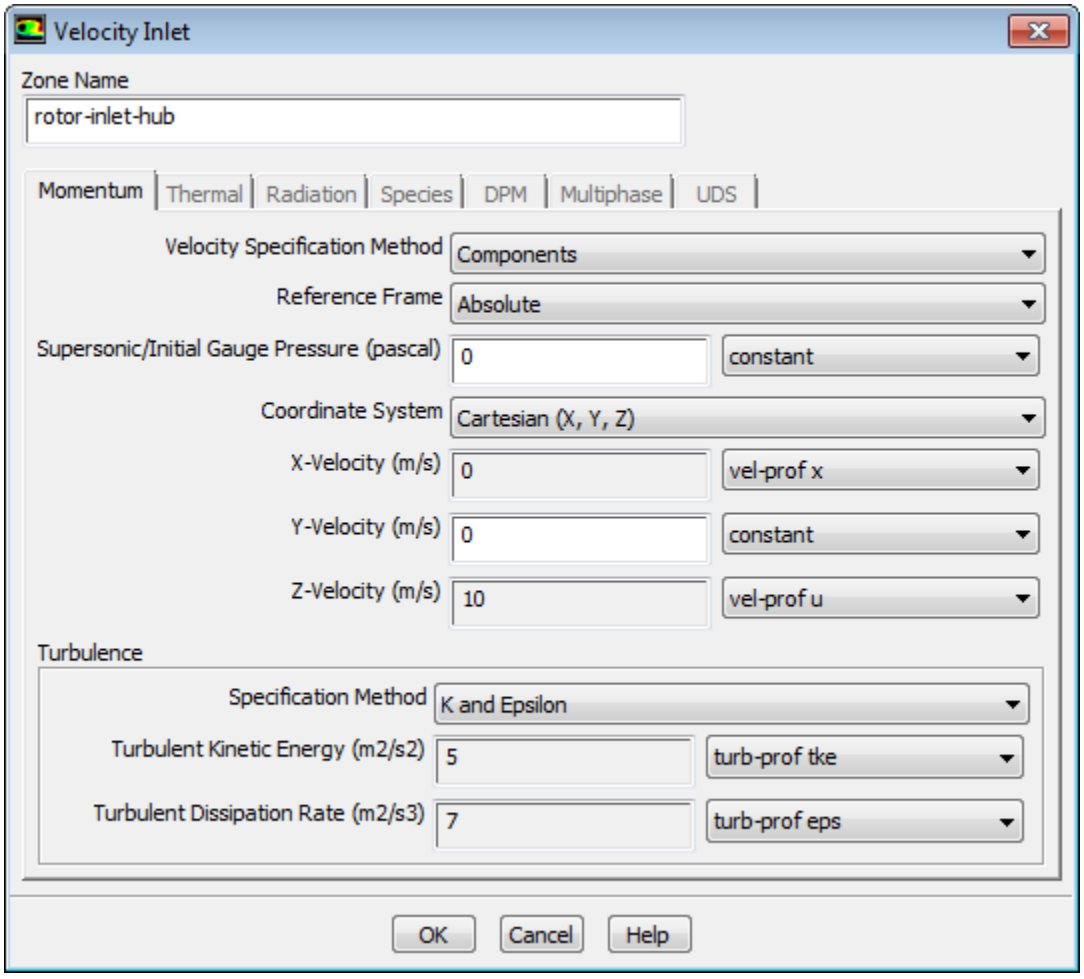
plotted. If the data plotted does not agree with your specified profile, this means that there is an error in the profile file.

- For 3D calculations, use the **Contours** dialog box to display contours on the appropriate boundary zone surface. The **Node Values** check button must be turned on in order for you to view the profile data. If the data shown in the contour plot does not agree with your specified profile, this means that there is an error in the profile file.

6.6.3.3. Example

For the example given in Example (p. 437), the profiles are used for inlet values of x velocity, turbulent kinetic energy, and turbulent kinetic energy dissipation rate, as illustrated in Figure 6.75: Example of Using Profiles as Boundary Conditions (p. 443). (The y velocity is set to a constant value of zero, since it is assumed negligible. However, a profile of y velocity could also be used.)

Figure 6.75: Example of Using Profiles as Boundary Conditions



6.6.4. Reorienting Profiles

For 3D cases only, ANSYS Fluent allows you to change the orientation of an existing profile so that it can be used at a boundary positioned arbitrarily in space. This allows you, for example, to take experimental data for an inlet with one orientation and apply it to an inlet in your model that has a different spatial orientation. Note that ANSYS Fluent assumes that the profile and the boundary are planar.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

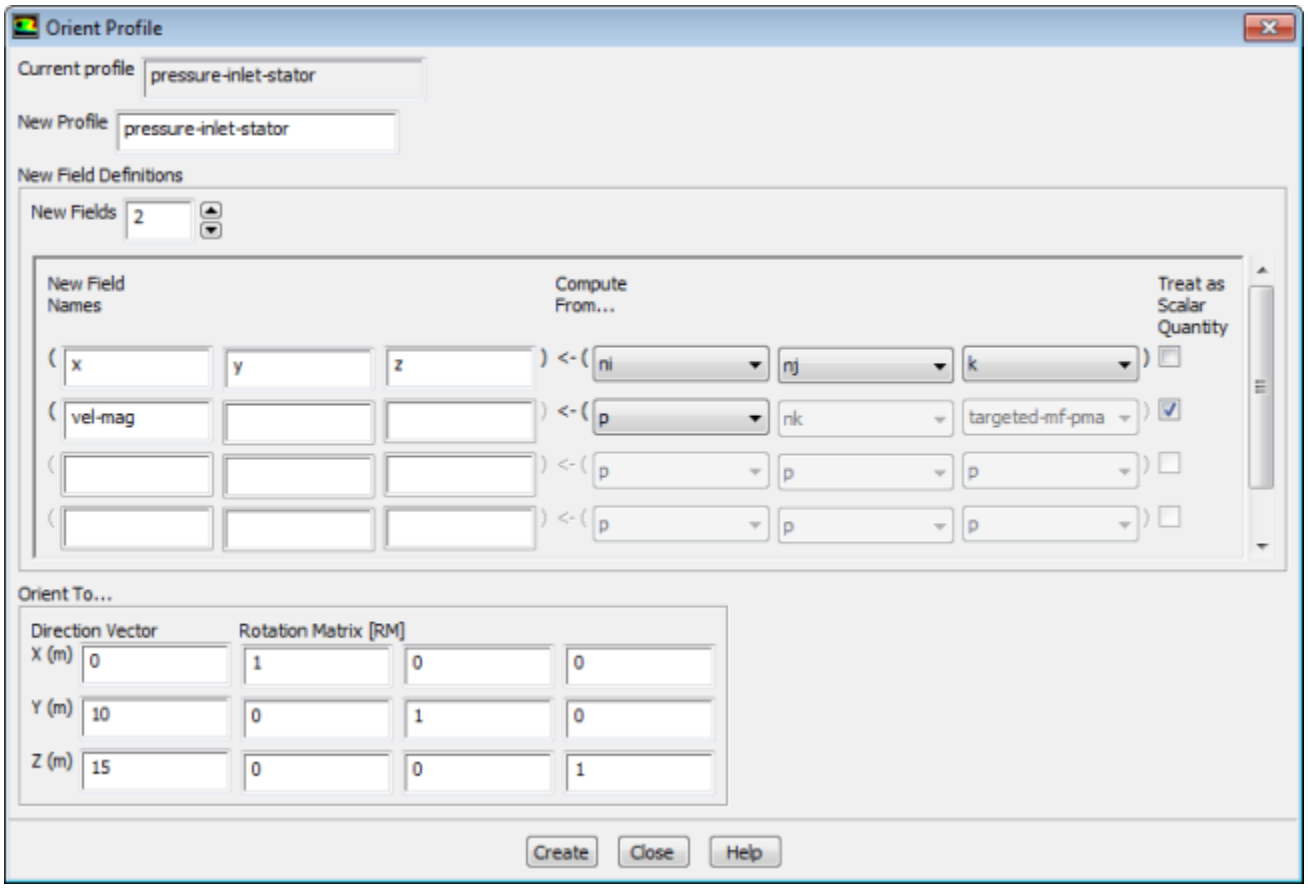
6.6.4.1. Steps for Changing the Profile Orientation

The procedure for orienting the profile data in the principal directions of a boundary is outlined below:

1. Define and read the profile as described in Using Profiles (p. 440).

2. In the Profiles Dialog Box (p. 2385), select the profile in the **Profile** list, and then click the **Orient...** button. This will open the Orient Profile Dialog Box (p. 2387) (Figure 6.76: The Orient Profile Dialog Box (p. 444)).

Figure 6.76: The Orient Profile Dialog Box



3. In the **Orient Profile** dialog box, enter the name of the new profile you want to create in the **New Profile** box.

4. Specify the number of fields you want to create using the up/down arrows next to the **New Fields** box. The number of new fields is equal to the number of vectors and scalars to be defined plus 1 (for the co-ordinates).

5. Define the coordinate field.

   a. Enter the names of the three coordinates (x, y, z) in the first row under **New Field Names**.

> **Important**
>
> Ensure that the coordinates are named x, y, and z only. Do not use any other names or upper case letters in this field.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Select the appropriate local coordinate fields for x, y, and z from the drop-down lists under **Compute From...**. (A selection of **0** indicates that the coordinate does not exist in the original profile; that is, the original profile was defined in 2D.)

6. Define the vector fields in the new profile.

   a. Enter the names of the 3 components in the directions of the coordinate axes of the boundary under **New Field Names**.

   Important

   Do not use upper case letters in these fields.

   b. Select the names of the 3 components of the vector in the local x, y, and z directions of the profile from the drop-down lists under **Compute From...**.

7. Define the scalar fields in the new profile.

   a. Enter the name of the scalar in the first column under **New Field Names**.

   Important

   Do not use upper case letters in these fields.

   b. Click the button under **Treat as Scalar Quantity** in the same row.

   c. Select the name of the scalar in the corresponding drop-down list under **Compute From...**.

8. Under **Orient To...**, specify the rotational matrix RM under the **Rotation Matrix [RM]**. The rotational matrix used here is based on Euler angles (γ, β, and α) that define an orthogonal system $x'y'z'$ as the result of the three successive rotations from the original system xyz. That is,

$$\begin{bmatrix} x' \\ y' \\ z' \end{bmatrix} = [RM] \begin{bmatrix} x \\ y \\ z \end{bmatrix} \tag{6.200}$$

$$RM = [C][B][A] \tag{6.201}$$

where C, B, and A are the successive rotations around the z, y, and x axes, respectively.

Rotation around the z axis:

$$C = \begin{bmatrix} \cos\gamma & -\sin\gamma & 0 \\ \sin\gamma & \cos\gamma & 0 \\ 0 & 0 & 1 \end{bmatrix} \tag{6.202}$$

Rotation around the y axis:

$$B = \begin{bmatrix} \cos\beta & 0 & \sin\beta \\ 0 & 1 & 0 \\ -\sin\beta & 0 & \cos\beta \end{bmatrix} \tag{6.203}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rotation around the x axis:

$$A=\begin{bmatrix}1 & 0 & 0\\ 0 & \cos\alpha & -\sin\alpha\\ 0 & \sin\alpha & \cos\alpha\end{bmatrix} \tag{6.204}$$

9. Under **Orient To...**, specify the **Direction Vector**. The **Direction Vector** is the vector that translates a profile to the new position, and is defined between the centers of the profile fields.

   Important

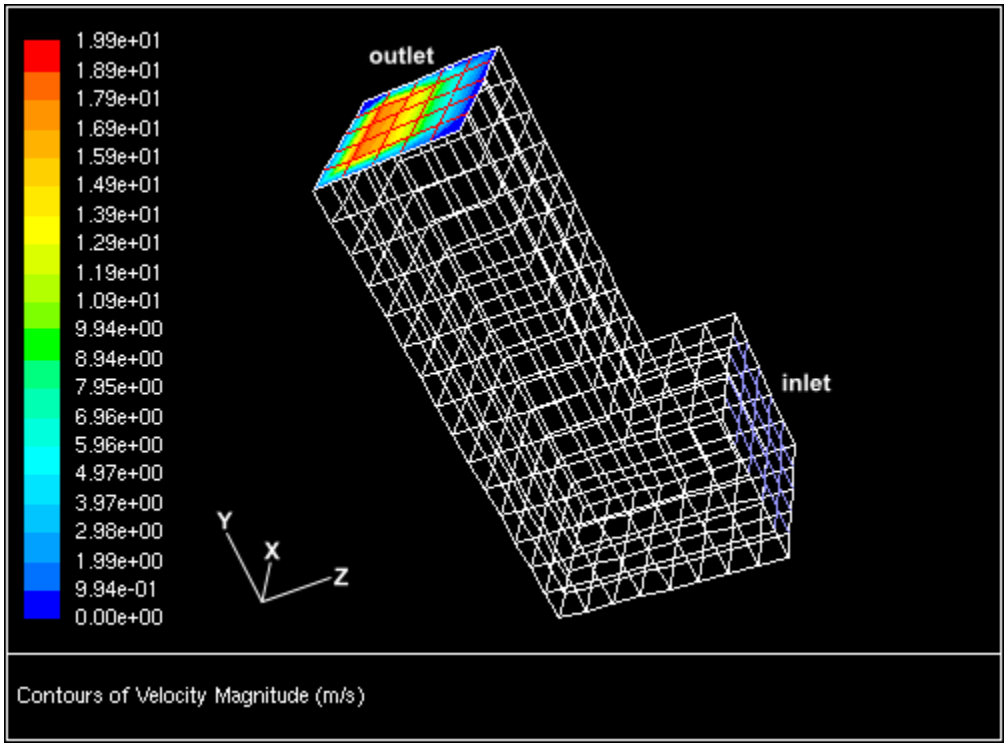
   Note that depending on your case, it may be necessary to perform only a rotation, only a translation, or a combination of a translation and a rotation.

10. Click the **Create** button in the **Orient Profile** dialog box, and your new profile will be created. Its name, which you entered in the **New Profile** box, will now appear in the **Profiles** dialog box and will be available for use at the desired boundary.

6.6.4.2. Profile Orienting Example

Consider the domain with a square inlet and outlet, shown in Figure 6.77: Scalar Profile at the Outlet (p. 446). A scalar profile at the outlet is written out to a profile file. The purpose of this example is to impose this outlet profile on the inlet boundary via a 90° rotation about the x axis. However, the rotation will locate the profile away from the inlet boundary. To align the profile to the inlet boundary, a translation via a directional vector must be performed.

Figure 6.77: Scalar Profile at the Outlet



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The problem is shown schematically in Figure 6.78: Problem Specification (p. 448). Φ_{out} is the scalar profile of the outlet. Φ'_{out} is the image of the Φ_{out} rotated 90° around the x axis. In this example, since $\gamma=\beta=0$, then $C=B=I$, where I is the identity matrix, and the rotation matrix is

$$RM=[C][B][A]=\begin{bmatrix}1 & 0 & 0\\ 0 & \cos 90^\circ & -\sin 90^\circ\\ 0 & \sin 90^\circ & \cos 90^\circ\end{bmatrix}=\begin{bmatrix}1 & 0 & 0\\ 0 & 0 & -1\\ 0 & 1 & 0\end{bmatrix} \tag{6.205}$$

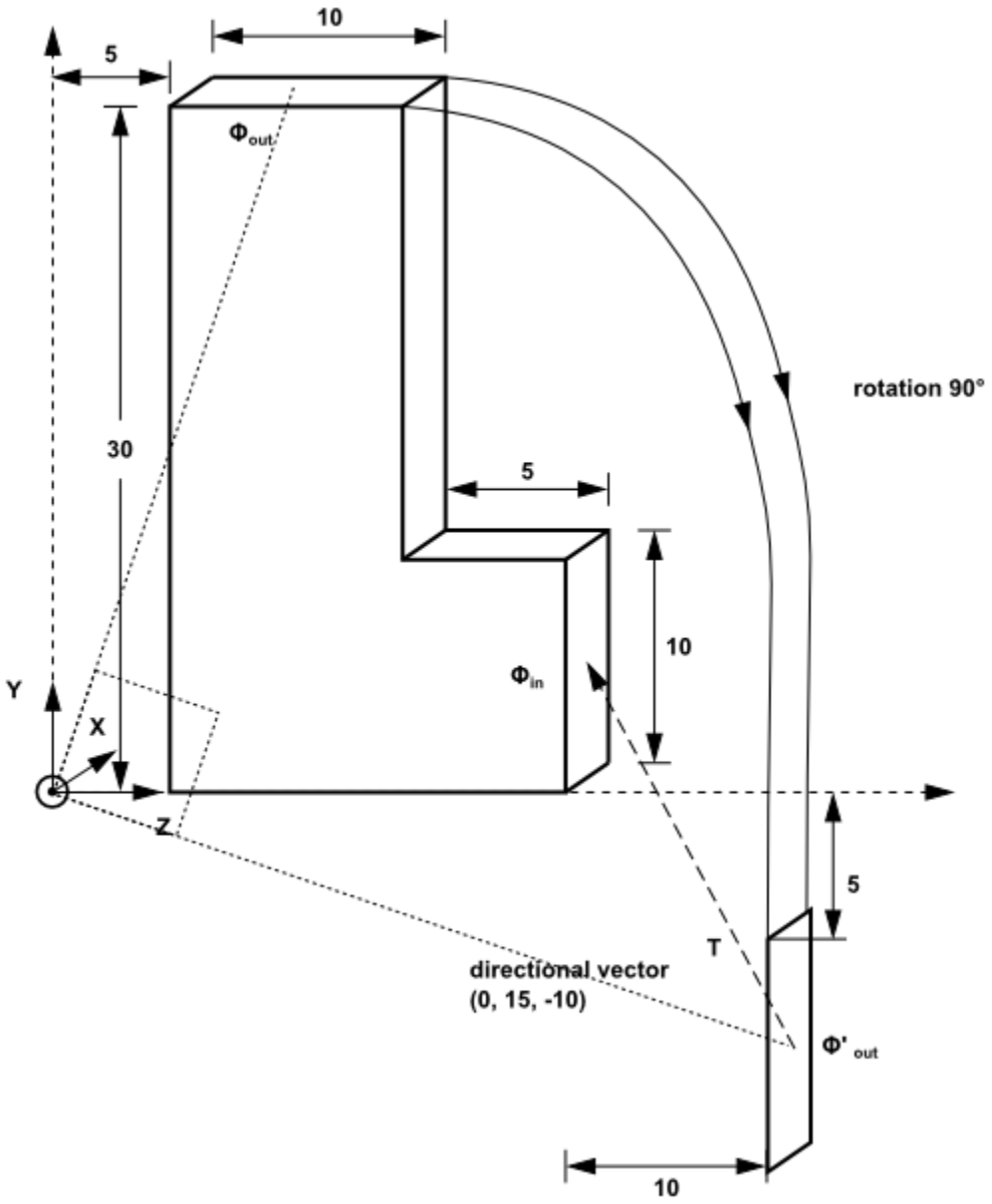
To overlay the outlet profile on the inlet boundary, a translation will be performed.

To overlay the outlet profile on the inlet boundary, a translation will be performed. The directional vector is the vector that translates Φ'_{out} to Φ_{in}. In this example, the directional vector is $(0,15,-10)^T$. The appropriate inputs for the **Orient Profile** dialog box are shown in Figure 6.76: The Orient Profile Dialog Box (p. 444).

Note that if the profile being imposed on the inlet boundary was due to a rotation of -90° about the x axis, then the rotational matrix RM must be found for $\gamma=\beta=0$ and $\alpha=-90^\circ$, and a new directional vector must be found to align the profile to the boundary.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.78: Problem Specification



6.6.5. Defining Transient Cell Zone and Boundary Conditions

There are three ways you can specify transient cell zone and boundary conditions:

- transient profile with a format similar to the standard profiles described in Profiles (p. 435)
- transient profile in a tabular format
- transient profile in comma separated values (CSV) format, as described in CSV Profiles (p. 438).

Important

For both methods, the cell zone or boundary condition will vary only in time; it must be spatially uniform. However, if the in-cylinder model is activated (In-Cylinder Settings (p. 690)), then you have the option to use the crank angle instead of time. Crank angles can be included

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in transient tables as well as transient profiles, in a similar fashion to time. Examples of transient profiles and transient tables in crank angle can be found in the sections that follow.

For information about boundary profiles, refer to Reading and Writing Profile Files (p. 53).

6.6.5.1. Standard Transient Profiles

The format of the standard transient profile file (based on the profiles described in Profiles (p. 435)) is

```
((profile-name transient n periodic?)
(field_name-1 a1 a2 a3 .... an)
(field_name-2 b1 b2 b3 .... bn)
.
.
.
.
(field_name-r r1 r2 r3 .... rn))
```

The profile name as well as the field names have to be shorter than 64 characters. One of the `field_name` s should be used for the `time` field, and the `time` field section *must* be in ascending order. n is the number of entries per field. The `periodic?` entry indicates whether or not the profile is time-periodic. Set it to `1` for a time-periodic profile, or `0` if the profile is not time-periodic.

An example is shown below:

```
((sampleprofile transient 3 0)
(time
1
2
3
)
(u
10
20
30
)
)
```

This example demonstrates the use of crank angle in a transient profile

```
((example transient 3 1)
(angle
0.000000e+00 1.800000e+02 3.600000e+02)
(temperature
3.000000e+02 5.000000e+02 3.000000e+02)
)
```

Important

All quantities, including coordinate values, must be specified in SI units because ANSYS Fluent does not perform unit conversion when reading profile files. Also, profile names must have all lowercase letters (for example, `name`). Uppercase letters in profile names are not acceptable.

You can read this file into ANSYS Fluent using the Profiles Dialog Box (p. 2385) or the **File/Read/Profile...** ribbon tab item.

Setup → Cell Zone Conditions → Profiles...

Setup → **Boundary Conditions** → **Profiles...**

File → **Read** → **Profile...**

See Using Profiles (p. 440) for details.

6.6.5.2. Tabular Transient Profiles

The format of the tabular transient profile file is

```
profile-name n_field n_data periodic?
field-name-1 field-name-2 field-name-3 .... field-name-n_field
v-1-1  v-2-1... ... ... ... v-n_field-1
v-1-2  v-2-2... ... ... ... v-n_field-2
.
.
.
.
.
v-1-n_data v-2-n_data ... ... ... ... v-n_field-n_data
```

The first field name (for example `field-name-1`) should be used for the `time` field, and the `time` field section, which represents the flow time, *must* be in ascending order. The `periodic?` entry indicates whether or not the profile is time-periodic. Set it to `1` for a time-periodic profile, or `0` if the profile is not time-periodic.

An example is shown below:

```
sampletabprofile 2 3 0
time u
1 10
2 20
3 30
```

This file defines the same transient profile as the standard profile example above.

If the periodicity is set to `1`, then `n_data` must be the number that closes one period.

An example is shown below:

```
periodtabprofile 2 4 1
time u
0 10
1 20
2 30
3 10
```

The following example uses crank angle instead of time:

```
example 2 3 1
angle temperature
0  300
180 500
360 300
```

Important

All quantities, including coordinate values, must be specified in SI units because ANSYS Fluent does not perform unit conversion when reading profile files. Also, profile names must have all lowercase letters (for example, `name`). Uppercase letters in profile names are not acceptable. When choosing the field names, spaces or parentheses should not be included.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can read this file into ANSYS Fluent using the `read-transient-table` text command.

`file` → `read-transient-table`

After reading the table into ANSYS Fluent, the profile will be listed in the Profiles Dialog Box (p. 2385) and can be used in the same way as a boundary profile. See Using Profiles (p. 440) for details.

6.6.5.3. Profiles for Moving and Deforming Meshes

You can use profiles to specify position, velocity, and angular velocity. There are two different valid profile formats for moving and deforming mesh cases.

```
profile-name n_field n_data periodic?
field-name-1 field-name-2 field-name-3 .... field-name-n_field
v-1-1  v-2-1... ... ... ... v-n_field-1
v-1-2  v-2-2... ... ... ... v-n_field-2
.
.
.
.
.
v-1-n_data v-2-n_data ... ... ... ... v-n_field-n_data
```

and

```
((profile-name transient n periodic?)
(field_name-1 a1 a2 a3 .... an)
(field_name-2 b1 b2 b3 .... bn)
.
.
.
.
(field_name-r r1 r2 r3 .... rn))
```

You must use the appropriate variable nomenclature when writing your profile so that Fluent can properly interpret your inputs. For position, use `x`, `y`, and `z`. For specifying velocity, use `v_x`, `v_y`, and `v_z`. Similarly, for angular velocity, use `omega_x`, `omega_y`, and `omega_z`.

6.7. Coupling Boundary Conditions with GT-POWER

GT-POWER users can define time-dependent boundary conditions in ANSYS Fluent based on information from GT-POWER. During the ANSYS Fluent simulation, ANSYS Fluent and GT-POWER are coupled together and information about the boundary conditions at each time step is transferred between them.

6.7.1. Requirements and Restrictions
6.7.2. User Inputs
6.7.3. Torque-Speed Coupling with GT-POWER

6.7.1. Requirements and Restrictions

Note the following requirements and restrictions for the GT-POWER coupling:

- The flow must be unsteady.
- The compressible ideal gas law must be used for density.
- Each boundary zone for which you plan to define conditions using GT-POWER must be a boundary of one of the following types:
  - velocity inlet

  - mass-flow inlet
  - pressure inlet
  - pressure outlet
  - wall (for details, see Torque-Speed Coupling with GT-POWER (p. 455))

  Also, a maximum of 20 boundary zones can be coupled to GT-POWER.

- Fluent checks the area of the 1D coupled boundary as calculated by Fluent and GT-POWER to ensure the calculated areas differ by no more than 5%. When the difference is more than 5%, Fluent stops with an error message and displays the two areas.
- If a mass-flow inlet or pressure inlet is coupled to GT-POWER, you must select **Normal to Boundary** as the **Direction Specification Method** in the **Mass-Flow Inlet** or **Pressure Inlet** dialog box. For a velocity inlet, you must select **Magnitude, Normal to Boundary** as the **Velocity Specification Method** in the **Velocity Inlet** dialog box.
- The mass flow specification method in the **Mass-Flow Inlet** boundary has to always be **Mass Flux** and not **Mass Flow Rate** when coupling with GTPower.
- Boundary conditions for the following variables can be obtained from GT-POWER:
  - velocity
  - temperature
  - pressure
  - density
  - species mass fractions
  - k and ε (Note that it is recommended that you define these conditions in ANSYS Fluent yourself, rather than using the data provided by GT-POWER, since the GT-POWER values are based on a 1D model.)
- Make sure that the material properties you set in ANSYS Fluent are the same as those used in GT-POWER, so that the boundary conditions will be valid for your coupled simulation.
- If your model includes species, make sure that the name of each species in GT-POWER corresponds to the **Chemical Formula** for that species material in the **Materials** dialog box. Also, recall that ANSYS Fluent can handle a maximum of 50 species.
- You can install the GT-POWER libraries in a directory other than the default location. If the GT-POWER libraries are loaded into a non-default location, you need to set the following environment variables:
  - `Fluent_GTIHOME` — the GTI installation directory where GT-POWER is installed
  - `Fluent_GTIVERSION` — the current version of the GTI installation

  Important

  GTI is not backwards compatible.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.7.2. User Inputs

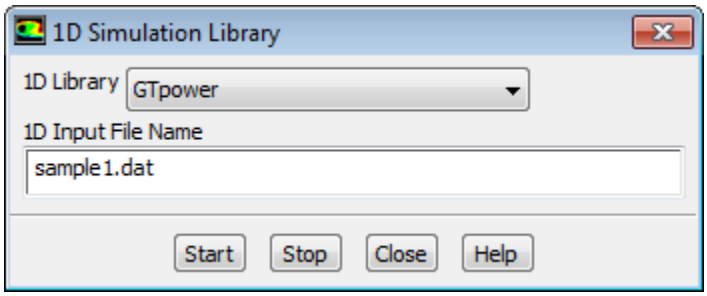
The procedure for setting up the GT-POWER coupling in ANSYS Fluent is presented below.

1. Read in the mesh file and define the models, materials, and boundary zone types (but *not* the actual boundary conditions), noting the requirements and restrictions listed in Requirements and Restrictions (p. 451).

2. Specify the location of the GT-POWER data and have ANSYS Fluent use them to generate user-defined functions for the relevant boundary conditions (using the 1D Simulation Library Dialog Box (p. 2662), shown in Figure 6.79: The 1D Simulation Library Dialog Box (p. 453)).

   User Defined → Model Specific → 1D Coupling...

   Figure 6.79: The 1D Simulation Library Dialog Box

   a. Select **GTpower** in the **1D Library** drop-down list.

   b. Specify the name of the GT-POWER input file in the **1D Input File Name** field.

   c. Click the **Start** button.

   When you click **Start**, GT-POWER will start up and ANSYS Fluent user-defined functions for each boundary in the input file will be generated.

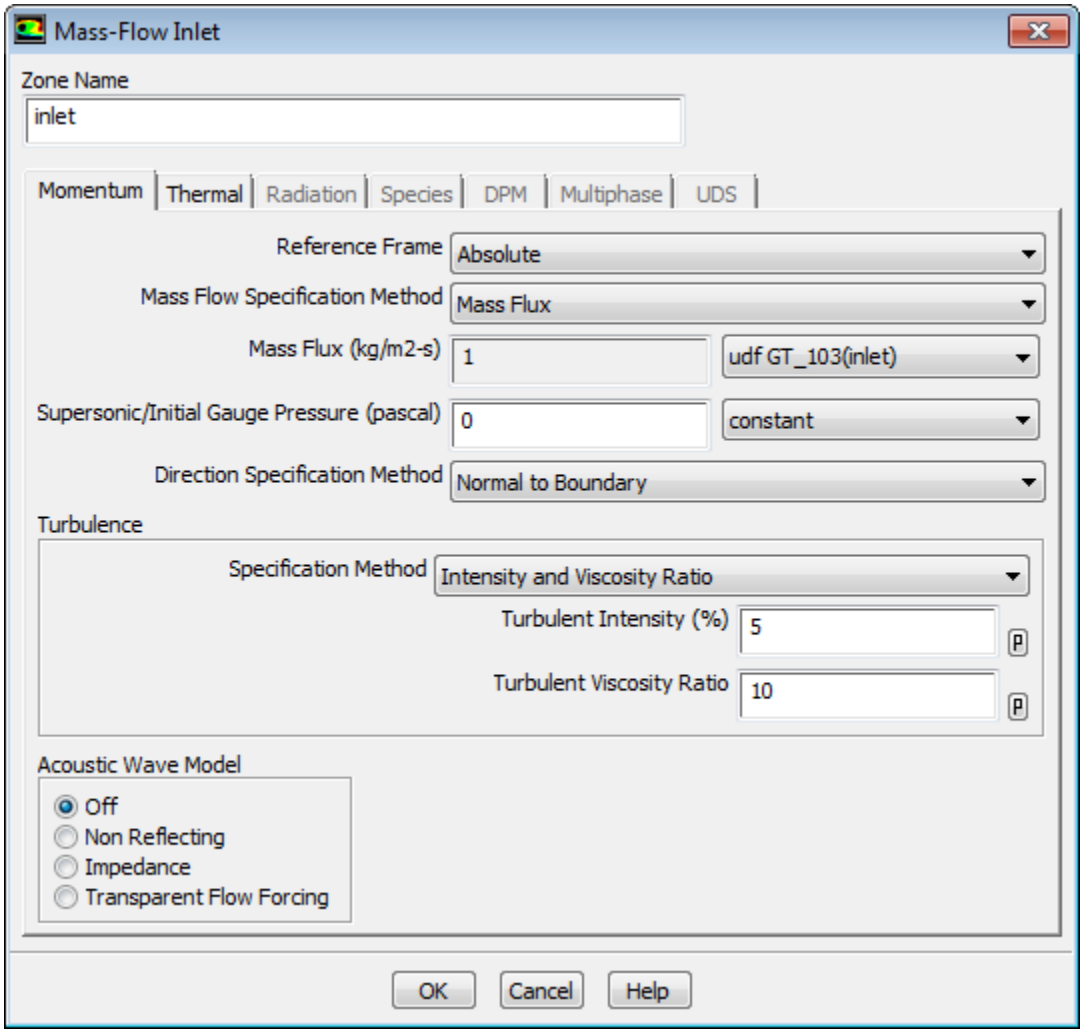
3. Set boundary conditions for all zones. For flow boundaries for which you are using GT-POWER data, select the appropriate UDFs as the conditions.

   Important

   Note that you must select the same UDF for all conditions at a particular boundary zone (as shown, for example, in Figure 6.80: Using GT-POWER Data for Boundary Conditions (p. 454)); this UDF contains all of the conditions at that boundary.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.80: Using GT-POWER Data for Boundary Conditions



4. If you plan to continue the simulation at a later time, starting from the final data file of the current simulation, specify how often you want to have the case and data files saved automatically.

   Solution → Calculation Activities (Autosave Case/Data) → Edit...

   To use a GT-POWER restart file to restart an ANSYS Fluent calculation, you must edit the GT-POWER input data file. See the GT-POWER User's Guide for instructions.

5. Continue the problem setup and calculate a solution in the usual manner.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.7.3. Torque-Speed Coupling with GT-POWER

Torque-speed coupling with GT-POWER is a unique case in which Fluent receives shaft speed from GT-POWER and returns torque. The procedure differs from the typical coupling between GT-Power and Fluent, and is as follows:

Important

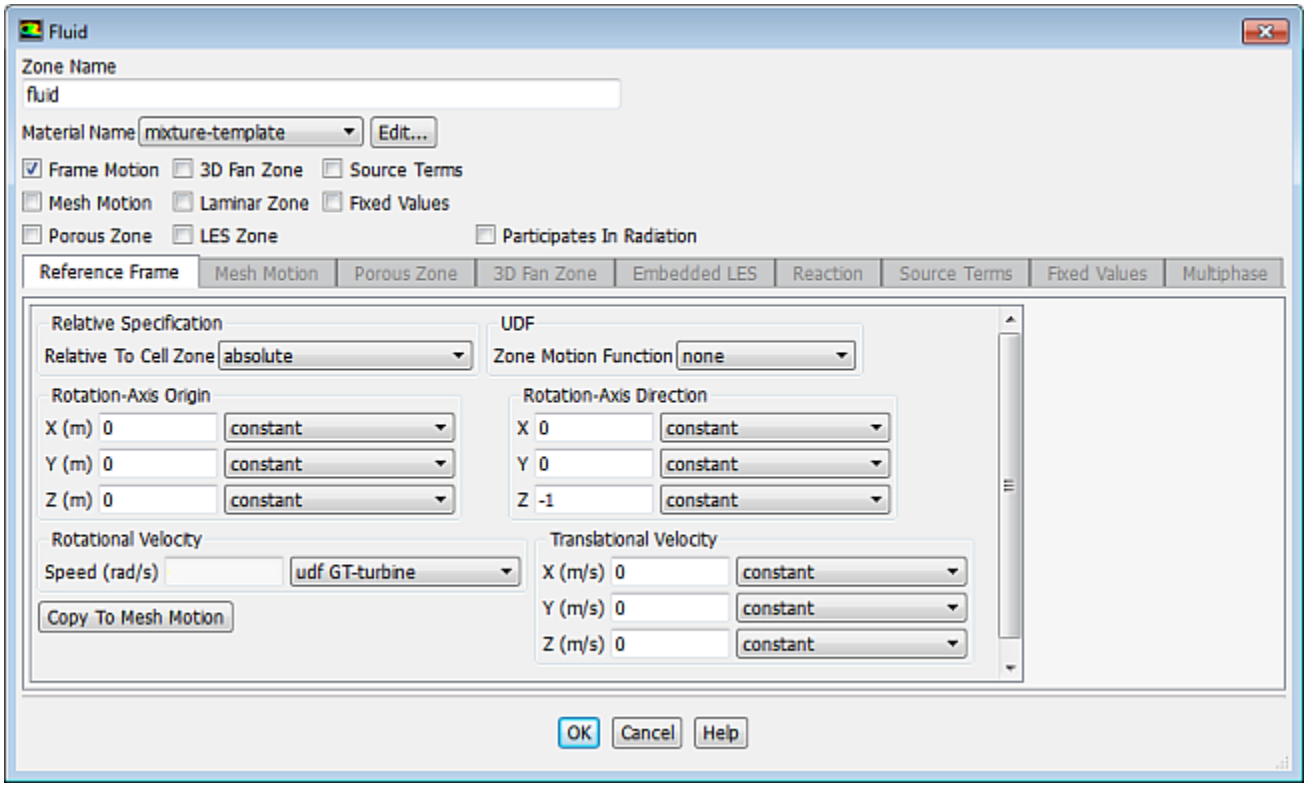
You must be using GT-POWER v2017.2 or greater to perform torque-speed coupling with ANSYS Fluent.

---

1. Complete steps 1 and 2 in the above section on User Inputs (p. 453). The `.dat` file should have at least one turbo-machine exposed to complete the coupling.

2. Enable **Frame Motion** in the cell zone that is receiving shaft speed from GT-Power. You must also select the appropriate UDF for **Rotational Velocity**. The UDF must correspond to the turbo-machine exposed from GT-POWER ("turbine", in the example shown below).

   Figure 6.81: Cell Zone Conditions for Torque-Speed Coupling with GT-POWER



3. Specify a **Wall** boundary on which the torque is calculated and transferred to GT-POWER. The following requirements apply:

   - The wall boundary must be attached to the rotating cell zone from step 2, to receive the shaft speed from GT-POWER.
   - The name of the wall boundary must be the same as the turbo-machine it is coupled with ("turbine", in the case presented above).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

During the solution, shaft speed is obtained from GT-POWER at every time-step, with Fluent calculating and returning torque within each time-step as well.

6.8. Coupling Boundary Conditions with WAVE

WAVE users can define time-dependent boundary conditions in ANSYS Fluent based on information from WAVE. During the ANSYS Fluent simulation, ANSYS Fluent and WAVE are coupled together and information about the boundary conditions at each time step is transferred between them.

6.8.1. Requirements and Restrictions
6.8.2. User Inputs

6.8.1. Requirements and Restrictions

Note the following requirements and restrictions for the WAVE coupling:

- WAVE must be installed and licensed.
- There are always five species that must be modeled in ANSYS Fluent just as they are defined in WAVE (F1, F2, F3, F4, and F5). It is recommended that realistic material properties be assigned to each of the five species.
- The flow must be unsteady.
- The compressible ideal gas law must be used for density.
- Each boundary zone for which you plan to define conditions using WAVE must be a flow boundary of one of the following types:
  - velocity inlet
  - mass-flow inlet
  - pressure inlet
  - pressure outlet

  Also, a maximum of 20 boundary zones can be coupled to WAVE.
- If a mass-flow inlet or pressure inlet is coupled to WAVE, you must select **Normal to Boundary** as the **Direction Specification Method** in the **Mass-Flow Inlet** or **Pressure Inlet** Dialog Box. For a velocity inlet, you must select **Magnitude, Normal to Boundary** as the **Velocity Specification Method** in the **Velocity Inlet** Dialog Box.
- Boundary conditions for the following variables can be obtained from WAVE:
  - velocity
  - temperature
  - pressure
  - density
  - species mass fractions

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - k and ε (Note that you are required to define these conditions in ANSYS Fluent yourself, since WAVE does not calculate them.)

- Make sure that the material properties you set in ANSYS Fluent are the same as those used in WAVE, so that the boundary conditions will be valid for your coupled simulation.

- If your model includes species, make sure that the name of each species in WAVE corresponds to the **Chemical Formula** for that species material in the **Create/Edit Materials** dialog box. Also, recall that ANSYS Fluent can handle a maximum of 50 species.

6.8.2. User Inputs

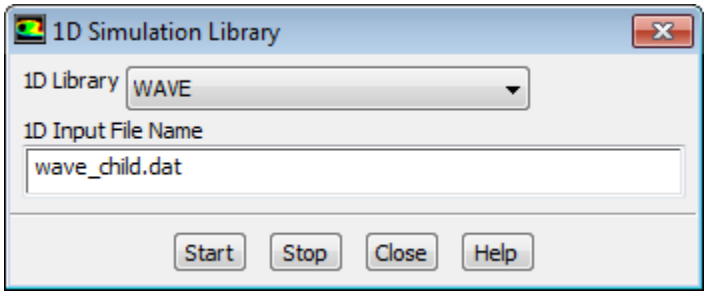
The procedure for setting up the WAVE coupling in ANSYS Fluent is presented below.

1. Read in the mesh file and define the models, materials, and boundary zone types.

2. Specify the location of the WAVE data and have ANSYS Fluent use them to generate user-defined functions for the relevant boundary conditions (using the 1D Simulation Library Dialog Box (p. 2662), shown in Figure 6.82: The 1D Simulation Library Dialog Box with WAVE Selected (p. 457)).

   User Defined → Model Specific → 1D Coupling...

   Figure 6.82: The 1D Simulation Library Dialog Box with WAVE Selected

   a. Select **WAVE** in the **1D Library** drop-down list.

   b. Specify the name of the WAVE input file in the **1D Input File Name** field.

   c. Click the **Start** button.

   When you click **Start**, WAVE will start up and ANSYS Fluent user-defined functions for each boundary in the input file will be generated.

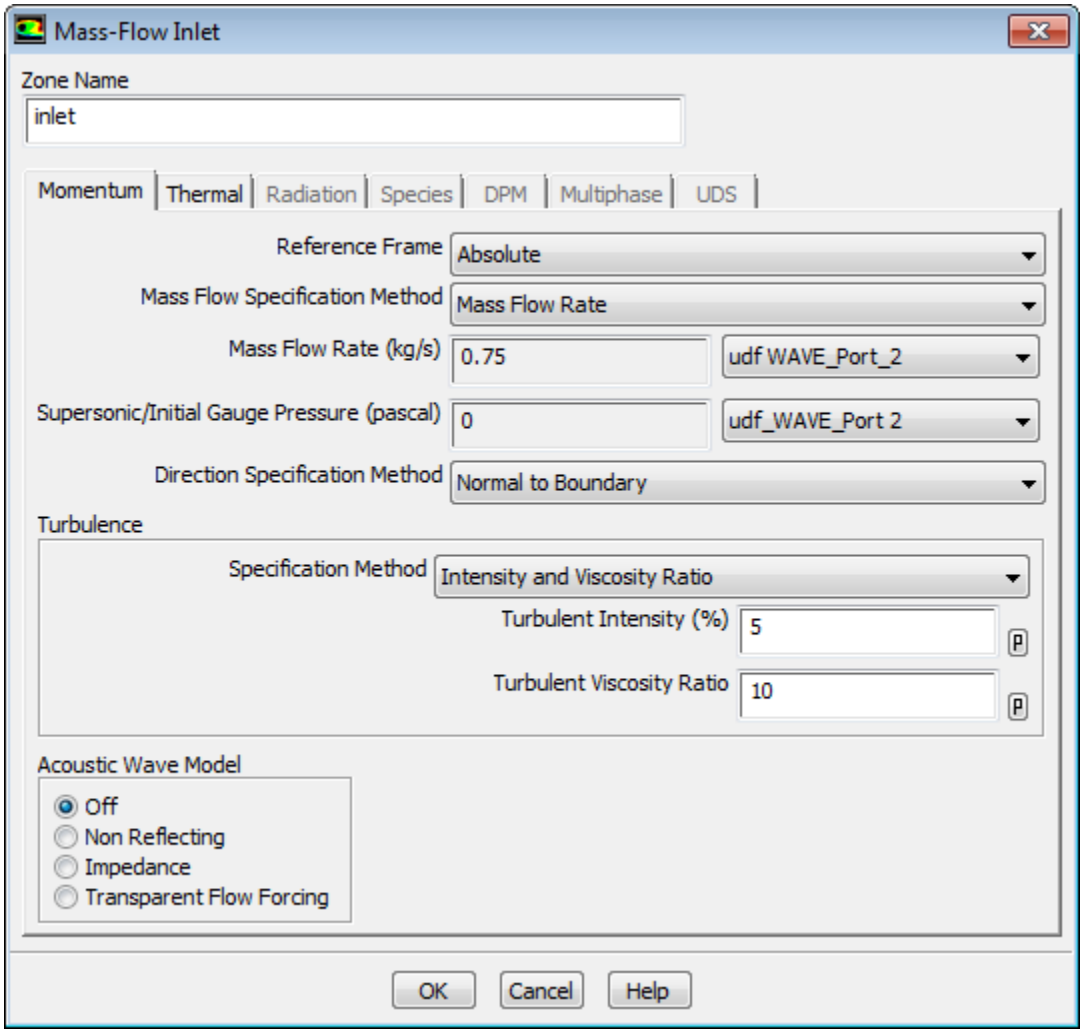
3. Set boundary conditions for all zones. For flow boundaries for which you are using WAVE data, select the appropriate UDFs as the conditions.

   Important

   Note that you must select the same UDF for all conditions at a particular boundary zone (as shown, for example, in Figure 6.83: Using WAVE Data for Boundary Conditions (p. 458)); this UDF contains all of the conditions at that boundary.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.83: Using WAVE Data for Boundary Conditions



4. If you plan to continue the simulation at a later time, restarting from the final data file of the current simulation, you need to instruct both ANSYS Fluent and WAVE how often that you want to automatically save your data. You should instruct ANSYS Fluent to automatically save case and data files at specified intervals using the autosave feature.

   Solution → Calculation Activities → Autosave On

   In addition, you should instruct WAVE as to how often it should generate its own restart files. See the WAVE User's Guide for instructions on this feature.

   Important

   To use the restart feature, the time interval for writing data files must be set to the same value in both ANSYS Fluent and WAVE. For example, if ANSYS Fluent has set the autosave feature to 100, then WAVE must also set the restart file write frequency to 100 as well.

5. Continue the problem setup and calculate a solution in the usual manner.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 7: Physical Properties

This chapter describes how to define materials, the physical equations used to compute material properties, and the methods you can use for each property input. Each property is described in detail in the following sections. If you are using one of the general multiphase models (VOF, mixture, or Eulerian), see Defining the Phases (p. 1377) for information about how to define the individual phases and their material properties.

7.1. Defining Materials
7.2. Defining Properties Using Temperature-Dependent Functions
7.3. Density
7.4. Viscosity
7.5. Thermal Conductivity
7.6. User-Defined Scalar (UDS) Diffusivity
7.7. Specific Heat Capacity
7.8. Radiation Properties
7.9. Mass Diffusion Coefficients
7.10. Standard State Enthalpies
7.11. Standard State Entropies
7.12. Unburnt Thermal Diffusivity
7.13. Kinetic Theory Parameters
7.14. Operating Pressure
7.15. Reference Pressure Location
7.16. Real Gas Models

7.1. Defining Materials

An important step in the setup of the model is to define the materials and their physical properties. Material properties are defined using either the Materials Task Page (p. 2300) or the **Materials** tree branch, where you can enter values for the properties that are relevant to the problem scope you have defined in the Models Task Page (p. 2169). These properties may include the following:

- density and/or molecular weights
- viscosity
- heat capacity
- thermal conductivity
- UDS diffusion coefficients
- mass diffusion coefficients
- standard state enthalpies
- kinetic theory parameters

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Properties may be temperature-dependent and/or composition-dependent, with temperature dependence based on a polynomial, piecewise-linear, or piecewise-polynomial function and individual component properties either defined by you or computed via kinetic theory.

The Materials Task Page (p. 2300) will show the properties that need to be defined for the active physical models. If any property you define requires the energy equation to be solved (for example, ideal gas law for density, temperature-dependent profile for viscosity), ANSYS Fluent will automatically enable the energy equation. Then you have to define the thermal boundary conditions and other parameters yourself.

For additional information, see the following sections:

7.1.1. Physical Properties for Solid Materials
7.1.2. Material Types and Databases
7.1.3. Using the Create/Edit Materials Dialog Box
7.1.4. Using a User-Defined Materials Database

7.1.1. Physical Properties for Solid Materials

For solid materials, only density, thermal conductivity, and heat capacity are defined. If you are modeling semi-transparent media, then radiation properties are also defined. You can specify a constant value, a temperature-dependent function, or a user-defined function for thermal conductivity; a constant value or temperature-dependent function for heat capacity; and a constant value for density.

If you are using the pressure-based solver, density and heat capacity for a solid material are not required unless you are modeling transient flow or moving solid zones. Heat capacity will appear in the list of solid properties for steady flows as well. The value will be used just for postprocessing enthalpy; not in the calculation.

7.1.2. Material Types and Databases

In ANSYS Fluent, you can define six types of materials: fluids, solids, mixtures, combusting-particles, droplet-particles, and inert-particles. Physical properties of fluids and solids are associated with named material; these materials are then assigned as boundary conditions for zones.

When you model species transport, define a mixture material, consisting of the various species involved in the problem. Properties will be defined for the mixture, as well as for the constituent species, which are fluid materials. The mixture material concept is discussed in detail in Mixture Materials (p. 959). Combusting-particles, droplet-particles, and inert-particles are available for the discrete-phase model, as described in The Concept of Discrete-Phase Materials (p. 1316).

ANSYS Fluent provides a built-in global database of approximately 675 predefined materials along with their properties and default values for each property. To define a material in the problem setup, you can copy materials from this global (site-wide) database and use the default properties or define new materials by editing their properties. The ANSYS Fluent materials database is located in the following file:

path `/ansys_inc/v181/fluent/fluent18.1.0/cortex/lib/propdb.scm`

where *path* is the directory in which you installed ANSYS Fluent.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In addition to using the ANSYS Fluent material database, you can also create your own database and materials, and use it to define the materials in your problem setup. See Using a User-Defined Materials Database (p. 467) for information about creating and using user-defined custom material databases.

Important

All the materials in your local materials list will be saved in the case file (when you write one). The materials specified by you will be available to you if you read this case file into a new solver session.

7.1.3. Using the Create/Edit Materials Dialog Box

The Create/Edit Materials Dialog Box (p. 2302) allows you to define the materials and their properties in your problem setup using either the **Fluent Database** or a **User-Defined Database**. It enables you to copy materials from a database, create new materials, and modify material properties.

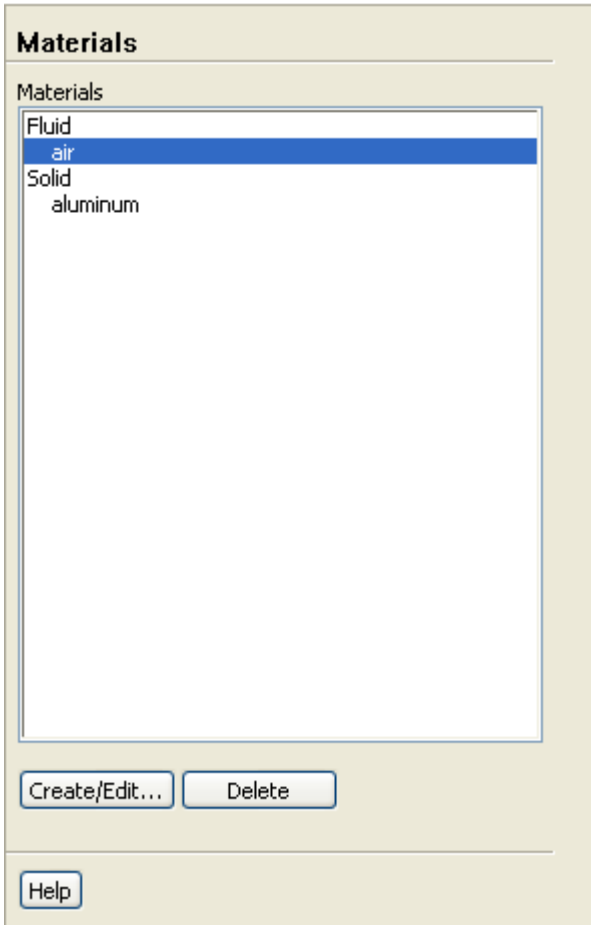
These generic functions are described in this section. The inputs for temperature-dependent properties are explained in Defining Properties Using Temperature-Dependent Functions (p. 475). The specific inputs for each material property are discussed in the remaining sections of this chapter.

The Create/Edit Materials Dialog Box (p. 2302) can be accessed in one of two ways:

- Through the **Materials** task page.

  Setup → **Materials**

Figure 7.1: The Materials Task Page

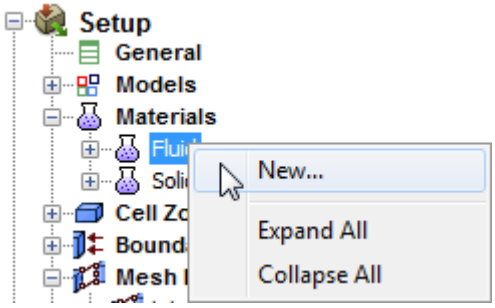


- Through the material types (**Fluid**, **Solid**, and so on) and defined in the case materials under **Materials** in the tree.

  For example, if you want to create a new fluid material you can access the **Create/Edit Materials** dialog box as follows:

  Setup → **Materials** → **Fluid** **New...**

Figure 7.2: The Materials Tree Branch



By default, your local materials list will include a single fluid material (air) and a single solid material (aluminum). If the fluid involved in your problem is air, you can use the default properties for air or

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

modify the properties. If the fluid in your problem is water, you can either copy water from the ANSYS Fluent database or create a new "water" material from scratch. If you copy water from the database, you can still make modifications to the properties of your local copy of water. See the following sections for detailed information on how to change material properties.

Mixture materials will not exist in your local list unless you have enabled species transport (see Modeling Species Transport and Finite-Rate Chemistry (p. 957)). Similarly, inert, droplet, and combusting particle materials will not be available unless you have created a discrete phase injection of these particle types (see Modeling Discrete Phase (p. 1235)). When a mixture material is copied from the database, all of its constituent fluid materials (species) will automatically be copied over as well.

7.1.3.1. Modifying Properties of an Existing Material

Probably, the most common operation you will perform in the Create/Edit Materials Dialog Box (p. 2302) is the modification of properties for an existing material. The steps for this procedure are as follows:

1. In the Materials Task Page (p. 2300), select the material you want to modify and click the **Create/Edit...** button. (You can also right-click the material in the tree and select **Edit...** from the material submenu.)

2. In the Create/Edit Materials Dialog Box (p. 2302) select the type of material (**fluid**, **solid**, and so on) in the **Material Type** drop-down list.

3. Choose the material for which you want to modify properties, in the **Fluent Fluid Materials** drop-down list, **Fluent Solid Materials** list, or other similarly named list. The name of the list will be the same as the material type you selected in the previous step.

4. Make the required changes to the properties listed in the **Properties** section of the dialog box. You can use the scroll bar to the right of the **Properties** section to scroll through the listed items.

5. Click the **Change/Create** button to change the properties of the selected material to your new property settings.

To change the properties of an additional material, repeat the process described above. Click the **Change/Create** button after making changes to the properties for each material.

7.1.3.2. Renaming an Existing Material

Each material is identified by a name and a chemical formula (if one exists). You can change the name of a material, but not its chemical formula (unless you are creating a new material). The procedure for renaming a material is as follows:

1. In the Materials Task Page (p. 2300), select the material you want to rename and click the **Create/Edit...** button. (You can also right-click the material in the tree and select **Edit...** from the material submenu.)

2. In the Create/Edit Materials Dialog Box (p. 2302), choose the material for which you want to modify properties, in the **Fluent Fluid Materials** list, **Fluent Solid Materials** list, or other similarly named list. The name of the list will be the same as the material type you selected in the previous step.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Enter the new name in the **Name** field at the top of the Create/Edit Materials Dialog Box (p. 2302).

   Important

   The maximum character length you can enter in the **Name** field is 29. If you enter a material name that is more than 29 characters long, ANSYS Fluent will print an error message in the console window.

4. Click the **Change/Create** button.

   A Question Dialog Box (p. 18) will appear, asking you if the original material should be overwritten.

   If you are renaming the original material, click **Yes** to overwrite it. If you were creating a new material, click **No** to retain the original material.

To rename another material, repeat the process described above. Click the **Change/Create** button after renaming each material.

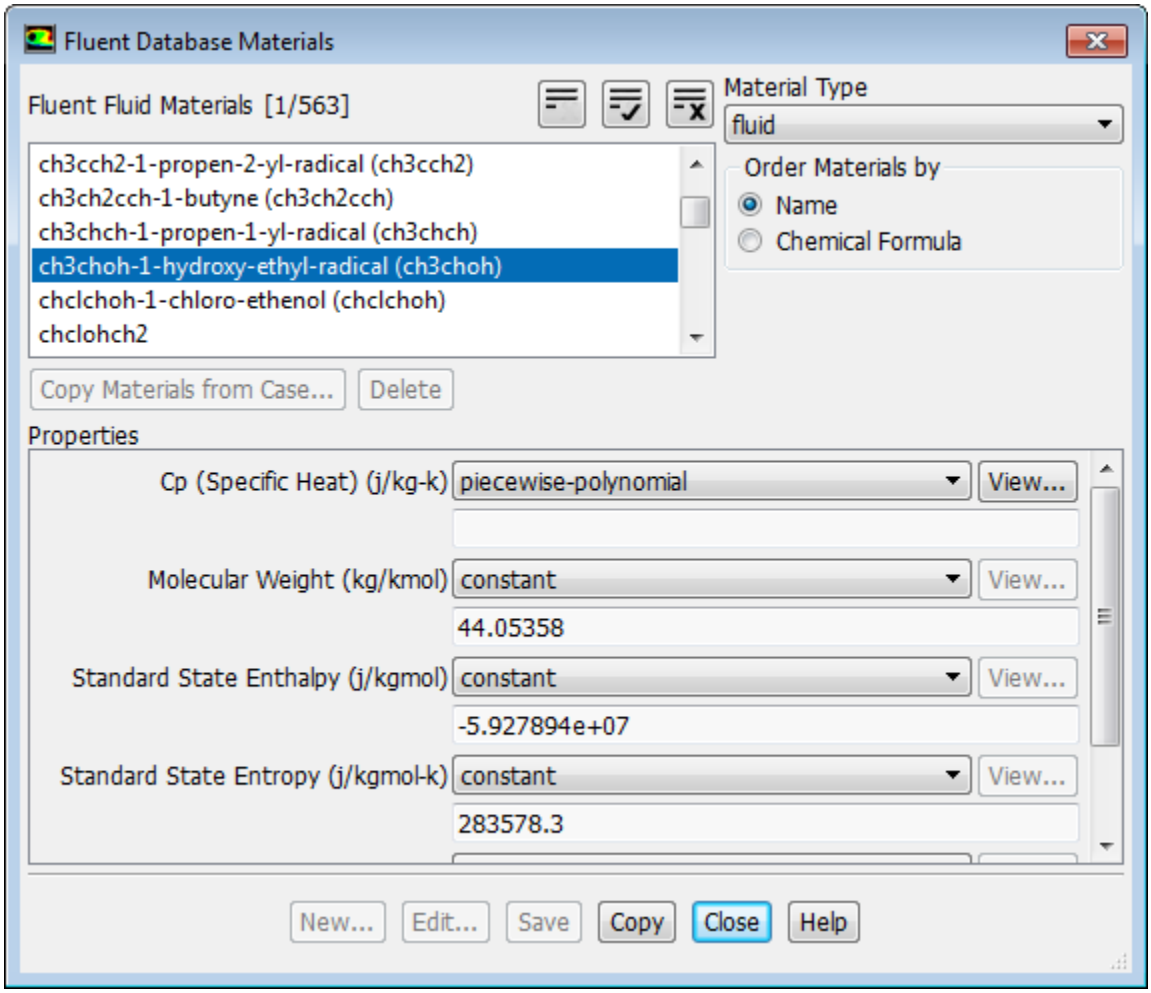
7.1.3.3. Copying Materials from the ANSYS Fluent Database

The global (site-wide) materials database contains many commonly used fluid, solid, and mixture materials, with property data from several different sources [66] (p. 2890), [95] (p. 2892), [129] (p. 2894), [45] (p. 2889). To use one of these materials in your problem, copy it from the ANSYS Fluent database to your local materials list. The procedure for copying a material is as follows:

1. Click the **Fluent Database...** button in the Create/Edit Materials Dialog Box (p. 2302) to open the Fluent Database Materials Dialog Box (p. 2312) (Figure 7.3: Fluent Database Materials Dialog Box (p. 465)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.3: Fluent Database Materials Dialog Box



2. Select the type of material (**fluid**, **solid**, and so on) in the **Material Type** drop-down list.

3. In the **Fluent Fluid Materials** list, **Fluent Solid Materials** list, or other similarly named list, choose the materials you want to copy by clicking on them. The properties of the selected material will be displayed in the **Properties** area.

4. To check the material properties, use the scroll bar to the right of the **Properties** area to scroll through the listed items. For some properties, temperature-dependent functions are available in addition to the constant values. Select one of the function types in the drop-down list to the right of the property and the relevant parameters will be displayed. You cannot edit these values, but the dialog boxes in which they are displayed function in the same way as those used for setting temperature-dependent property functions (Defining Properties Using Temperature-Dependent Functions (p. 475)).

   The inactive buttons in the Fluent Database Materials Dialog Box (p. 2312) are operations that are applicable only for a user-defined database. These operations will be available when you click the ***User-Defined Database...*** *button in the Create/Edit Materials Dialog Box (p. 2302).*

5. Click **Copy**.

   The materials and their properties will be downloaded from the database into your local list, and your copy of properties will now be displayed in the Materials Task Page (p. 2300) and in the tree under the appropriate material type.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Close the Fluent Database Materials Dialog Box (p. 2312).

After copying a material from the database, you can modify its properties or change its name, as described earlier in this section. The original material in the database will not be affected by any changes made to your local copy of the material.

7.1.3.4. Creating a New Material

If the material you want to use is not available in the database, you can easily create a new material for the local list. This material will be available for use only for the current problem and will not be saved in the ANSYS Fluent database. The procedure for creating a new material is as follows:

1. In the Materials Task Page (p. 2300), click the **Create/Edit...** button. (You can also right-click the corresponding material type in the tree (**Fluid**, **Solid**, and so on) and select **New...** from the material type submenu.)

2. Select the new material type (**fluid**, **solid**, and so on) in the **Material Type** drop-down list. It does not matter which material is selected in the **Fluent Fluid Materials**, **Fluent Solid Materials**, or other similarly named list.

3. Enter the new material name in the **Name** field.

   Important

   The maximum character length you can enter in the **Name** field is 29. If you enter a material name that is more than 29 characters long, ANSYS Fluent will print an error message in the console window.

4. Set the material's properties in the **Properties** area. If there are many properties listed, you may use the scroll bar to the right of the **Properties** area to scroll through the listed items.

5. Click the **Change/Create** button. A Question Dialog Box (p. 18) will appear, asking you if the original material should be overwritten.

   a. Click **No** to retain the original material and *add* your new material to the list. A dialog box will appear asking you to enter the chemical formula of your new material.

   b. Click **OK**, enter the formula if it is known. Else, leave the formula blank and click **OK**. Select the **Change/Create** button and answer the **Question**.

      The Materials Task Page (p. 2300) and the **Material** tree branch will be updated to show the new material name and chemical formula under the appropriate material type (**Fluid**, **Solid**, or others).

7.1.3.5. Saving Materials and Properties

All the materials and properties in your local list are saved in the case file when it is written. If you read this case file into a new solver session, all of your materials and properties will be available for use in the new session.

7.1.3.6. Deleting a Material

If there are materials list that you no longer need, you can delete them in one of the following ways:

- Using the Materials Task Page (p. 2300).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the Materials Task Page (p. 2300), select the material to be deleted and click the **Delete** button at the bottom of the Materials Task Page (p. 2300).

- Using the tree.

  Right-click the material in the tree and select **Delete** from the material submenu.

- Using the Create/Edit Materials Dialog Box (p. 2302).

  1. From the **Material Type** drop-down list, select the type of material (**fluid**, **solid**, and so on).
  2. Select the material to be deleted from the **Fluent Fluid Materials** drop-down list, **Fluent Solid Materials** list, or other similarly named list. The list's name will be the same as the material type you selected in the previous step.
  3. Click the **Delete** button.

Deleting materials from your local list will have no effect on the materials contained in the global database.

7.1.3.7. Changing the Order of the Materials List

By default, the materials in your local list and those in the database are listed alphabetically by name (for example, **air**, **atomic-oxygen (o)**, **carbon-dioxide (co2)**). If you prefer to list them alphabetically by chemical formula, select the **Chemical Formula** option under **Order Materials By**. The example materials listed, will now be in the order of: **air**, **co2 (carbon-dioxide)**, **o (atomic-oxygen)**. To change back to the alphabetical listing by name, choose the **Name** option under **Order Materials By**.

You may specify the ordering method separately for the Create/Edit Materials Dialog Box (p. 2302) and Fluent Database Materials Dialog Box (p. 2312). For example, you can order the database materials list by name. Each dialog box has its own **Order Materials By** options.

7.1.4. Using a User-Defined Materials Database

In addition to the Fluent Database Materials Dialog Box (p. 2312), you can also use or create a user-defined materials database using the User-Defined Database Materials Dialog Box (p. 2313). You can browse and do the following:

- select from existing user-defined databases
- copy materials from a user-defined database
- create a new database, create new materials
- add them to the user-defined database
- delete materials from the database
- copy materials from a case to a user-defined database
- view the database

The following sections will address each of these functionalities in detail.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.1.4.1. Opening a User-Defined Database

In ANSYS Fluent, you can open databases of custom materials saved as `.scm` files and use them to define the materials in your problem setup. The material data must be prepared in a specific format as shown in the examples that follow.

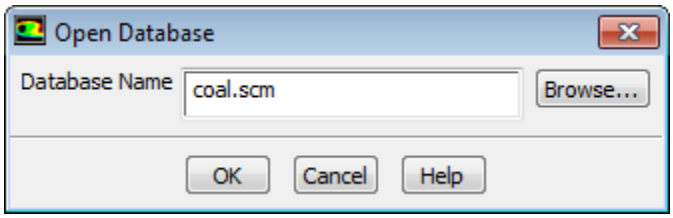
Examples:

The prescribed format for saving material properties information is shown here for air and aluminum. These files can be created in a text editor and saved with a `.scm` extension.

```
((air
fluid
(chemical-formula . #f)
(density (constant . 1.225)
(premixed-combustion 1.225 300))
(specific-heat (constant . 1006.43))
(thermal-conductivity (constant . 0.0242))
(viscosity (constant . 1.7894e-05)
(sutherland 1.7894e-05 273.11 110.56)
(power-law 1.7894e-05 273.11 0.666))
(molecular-weight (constant . 28.966))
)
(aluminum
solid
(chemical-formula . al)
(density (constant . 2719))
(specific-heat (constant . 871))
(thermal-conductivity (constant . 202.4))
(formation-entropy (constant . 164448.08))
))
```

To select a user-defined database, click the **User-Defined Database...** button in the Create/Edit Materials Dialog Box (p. 2302). This will open the Open Database Dialog Box (p. 2313).

Figure 7.4: Open Database Dialog Box



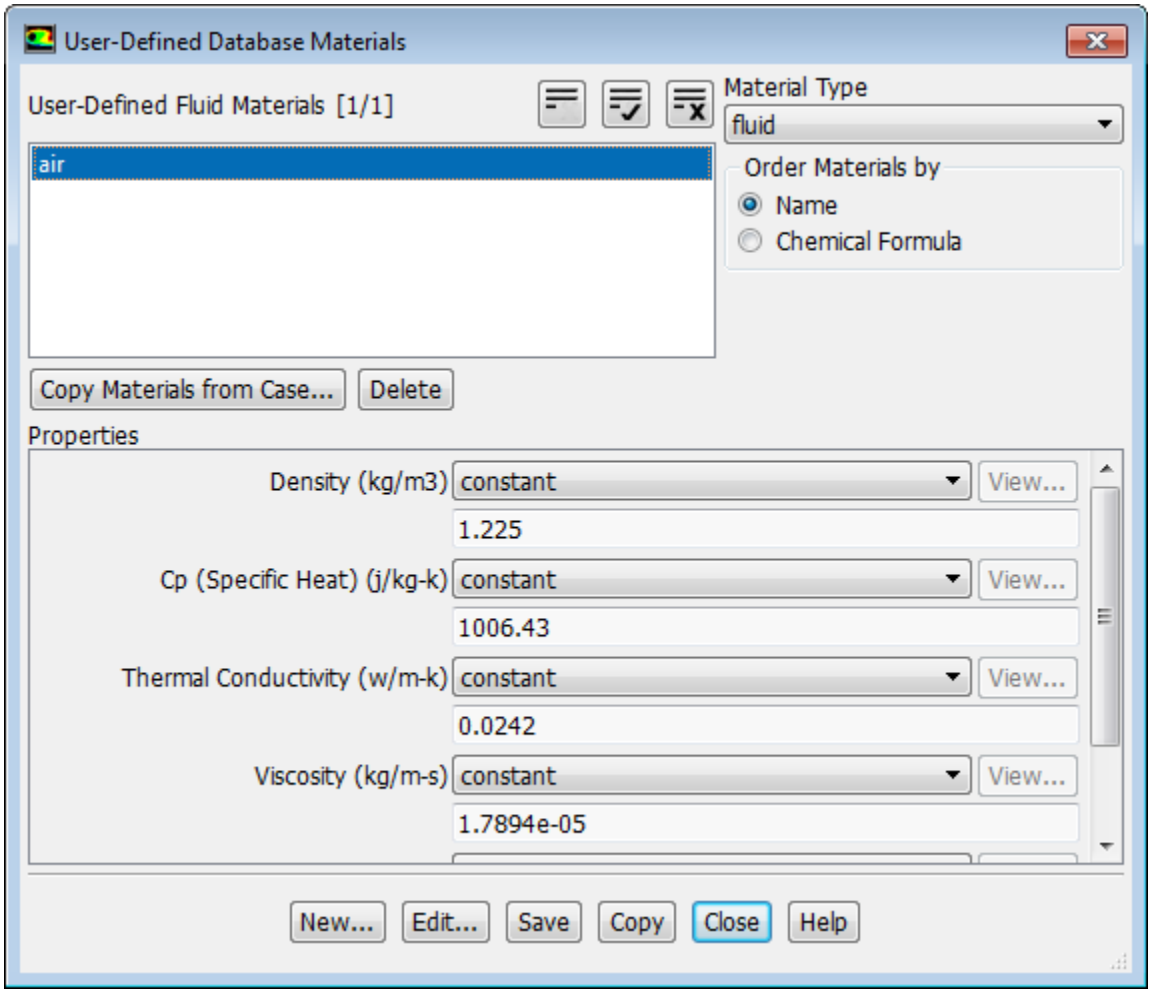
Click the **Browse...** button, select the database in The Select File Dialog Box (p. 19) that opens and click **OK**. Click **OK** in the Open Database Dialog Box (p. 2313) to open the User-Defined Database Materials Dialog Box (p. 2313).

7.1.4.2. Viewing Materials in a User-Defined Database

When an existing user-defined database is opened, the materials present in the database are listed in the User-Defined Database Materials Dialog Box (p. 2313). You can select the material type in the **Material Type** drop-down list and the corresponding materials will appear in the **User-Defined Fluid Materials**, **User-Defined Solid Materials** or other similarly named list. The name of the list will be the same as the material type you selected.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.5: User-Defined Database Materials Dialog Box



The properties of the selected material will appear in the **Properties** section of the dialog box. This dialog box is similar to the Fluent Database Materials Dialog Box (p. 2312) in function and operation.

7.1.4.3. Copying Materials from a User-Defined Database

The procedure for copying a material from a custom database is as follows:

1. In the Create/Edit Materials Dialog Box (p. 2302), click the **User-Defined Database...** button and open the database from which you want to copy the material.

2. In the User-Defined Database Materials Dialog Box (p. 2313) of the selected database, select the type of material (**fluid**, **solid**, and so on) in the **Material Type** drop-down list.

3. In the **User-Defined Fluid Materials** list, **User-Defined Solid Materials** list, or other similarly named list (the list's name will be the same as the material type you selected in the previous step), choose the materials you want to copy by clicking on them. The properties are displayed in the **Properties** area.

4. If you want to check the material properties, use the scroll bar to the right of the **Properties** area to scroll through the listed items.

5. Click the **Copy** button.

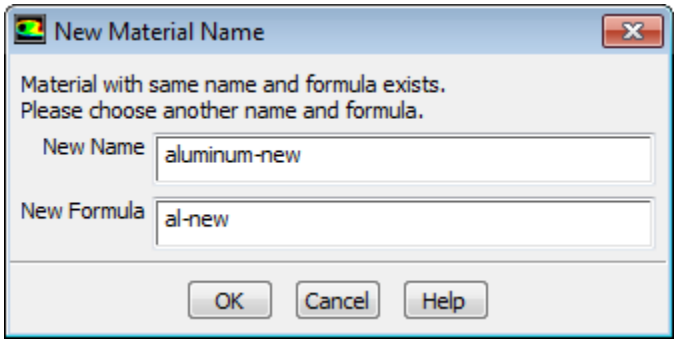
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The selected materials and their properties will be copied from the database into your local list, and your copy of the properties will now be displayed in the Materials Task Page (p. 2300) and under the **Materials** tree branch.

To copy all the materials from the database in one step, click the button (*) next to **User-Defined Materials** title and click **Copy**.*

If a material with the same name is already defined in the case, ANSYS Fluent will prompt you to enter a new name and formula in the New Material Name Dialog Box (p. 2317). Enter a new name and formula in the respective fields and click **OK** to make a local copy of the material.

Figure 7.6: New Material Name Dialog Box



6. Close the User-Defined Database Materials Dialog Box (p. 2313).

After copying a material from the database, you may modify its properties or change its name, as described earlier in Using the **Create/Edit Materials** Dialog Box (p. 461). The material in the database will not be affected by any changes you make to your local copy of the material.

7.1.4.4. Copying Materials from the Case to a User-Defined Database

The procedure for copying a material to a custom database is as follows:

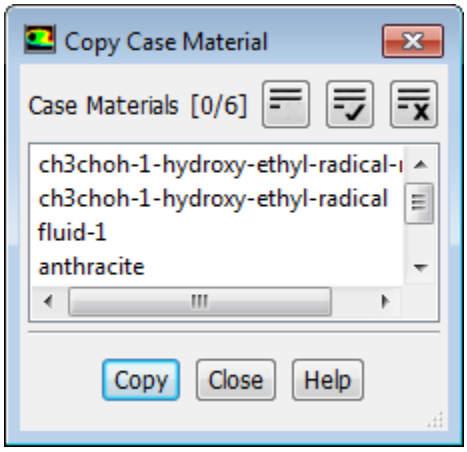
1. In the Create/Edit Materials Dialog Box (p. 2302), click **User-Defined Database...**.

2. In the Open Database Dialog Box (p. 2313), select the database to which you want to copy the material. If you want to create a new database, enter the name of the new database in the **Database Name** field and click **OK**. A Question Dialog Box (p. 18) will ask you to confirm if you want to create a new file. Click **Yes** to confirm.

3. In the User-Defined Database Materials Dialog Box (p. 2313), click **Copy Materials From Case....** This will open the Copy Case Material Dialog Box (p. 2315).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.7: Copy Case Material Dialog Box



a. In the Copy Case Material Dialog Box (p. 2315), select the materials that you want to copy.

   *To select all the materials, click the shaded icon to the right of the **Case Materials** title. Clicking on the unshaded icon will deselect the selections in the list.*

b. Click **Copy** and close the dialog box.

> **Note**
>
> Do not copy materials one by one. This will result in previously copied materials getting overwritten by the new ones. Instead, select all the materials to be copied and click **Copy**.

7.1.4.5. Modifying Properties of an Existing Material

You can modify the properties of an existing material and use the modified material in the problem setup and save the modified material to the materials database.

1. In the Create/Edit Materials Dialog Box (p. 2302), click the **User-Defined Database...** button and open the database that you want to use.

   a. In the User-Defined Database Materials Dialog Box (p. 2313) of the selected database, select the type of material (**fluid**, **solid**, and so on) in the **Material Type** drop-down list.

   b. In the **User-Defined Fluid Materials** list, **User-Defined Solid Materials** list, or other similarly named list (the name of the list will be the same as the material type you selected in the previous step), select the material to be modified.

   c. Click **Edit...** to open the Material Properties Dialog Box (p. 2315).

      i. In the **Materials Properties** list, select the property to be modified and click **Edit...** to open the Edit Property Methods Dialog Box (p. 2316).

      ii. Select the method to be modified in the **Material Properties** list of the Edit Property Methods Dialog Box (p. 2316) and click **Edit...** under **Edit Properties**, in order to modify the properties.

      iii. Make the changes in the corresponding method dialog box and click **OK**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

iv. Click **Apply** in the Material Properties Dialog Box (p. 2315).

d. To use the modified material in the problem setup, click **Copy** in the User-Defined Database Materials Dialog Box (p. 2313).

e. To save the modified material to the database, click **Save** and close the dialog box.

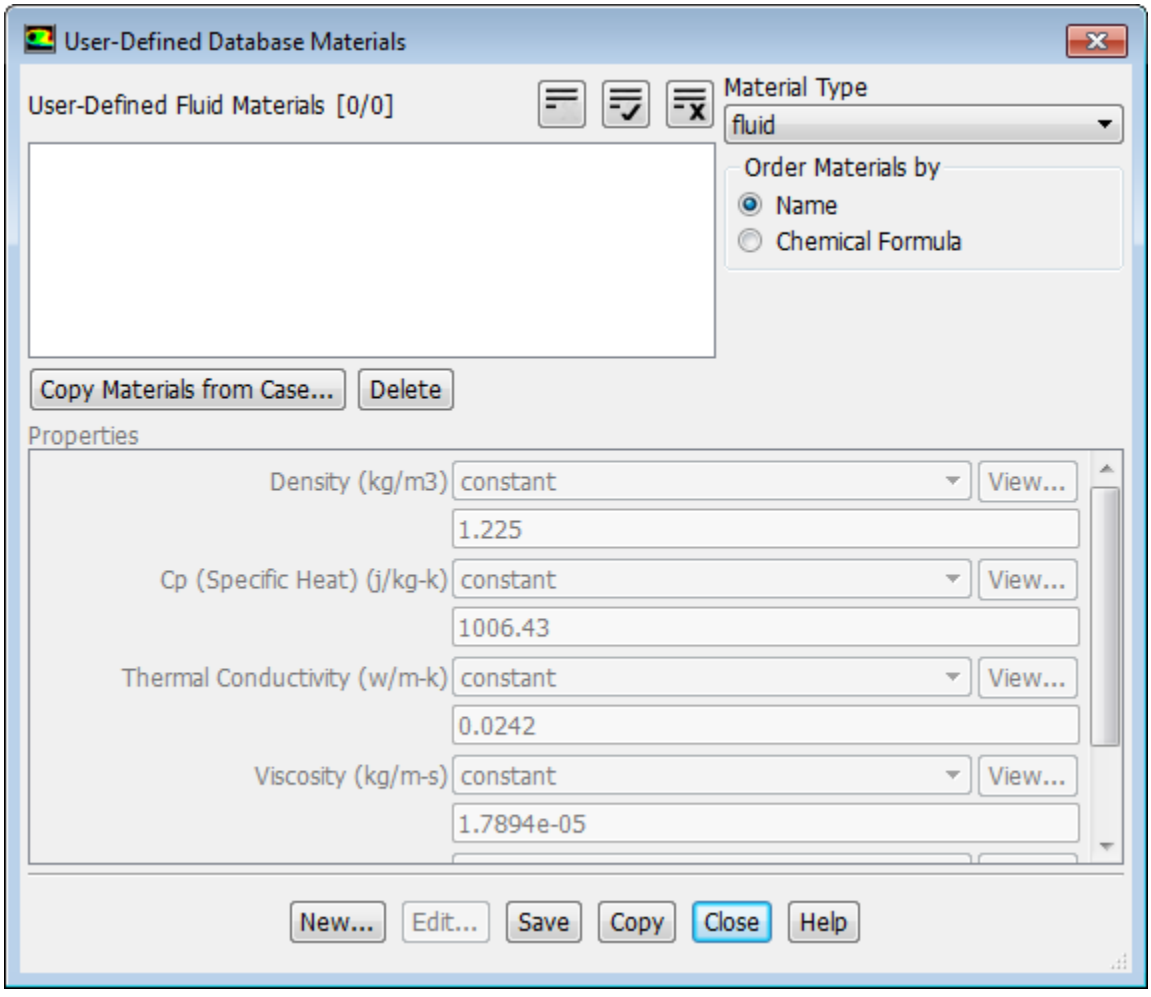
7.1.4.6. Creating a New Materials Database and Materials

Using the Create/Edit Materials Dialog Box (p. 2302), you can create a new materials database, copy materials to this database, and also create new materials from scratch. The procedure for creating a new database and add new materials to the database is as follows:

1. In the Materials Task Page (p. 2300), click **User-Defined Database...**.
2. In the Open Database Dialog Box (p. 2313), enter the name of the database that you are creating and click **OK**.
3. A dialog box will appear asking you confirm the creation of a new file. Click **Yes** to confirm.

   This will open a blank User-Defined Database Materials Dialog Box (p. 2313) (Figure 7.8: User-Defined Database Materials Dialog Box: Blank (p. 472)).

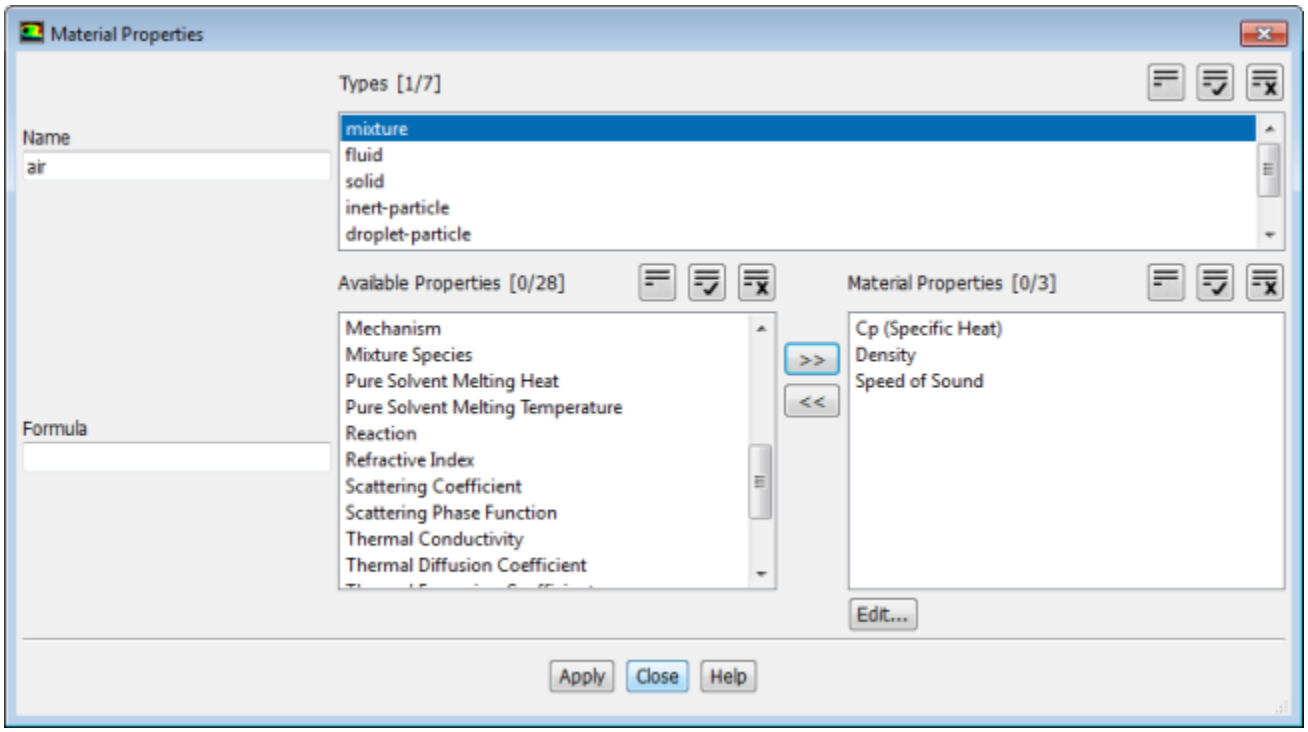
Figure 7.8: User-Defined Database Materials Dialog Box: Blank



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click **New...** in the User-Defined Database Materials Dialog Box (p. 2313). This will open a blank **Material Properties** dialog box.

Figure 7.9: Material Properties Dialog Box: Blank



a. In the Material Properties Dialog Box (p. 2315), under **Types**, select the material type. You can select from **fluid**, **solid**, **inert-particle**, **droplet-particle**, **combusting-particle**, and **mixture** materials.

b. Enter the name and formula (if required) of the material that you are creating in the **Name** and **Formula** fields.

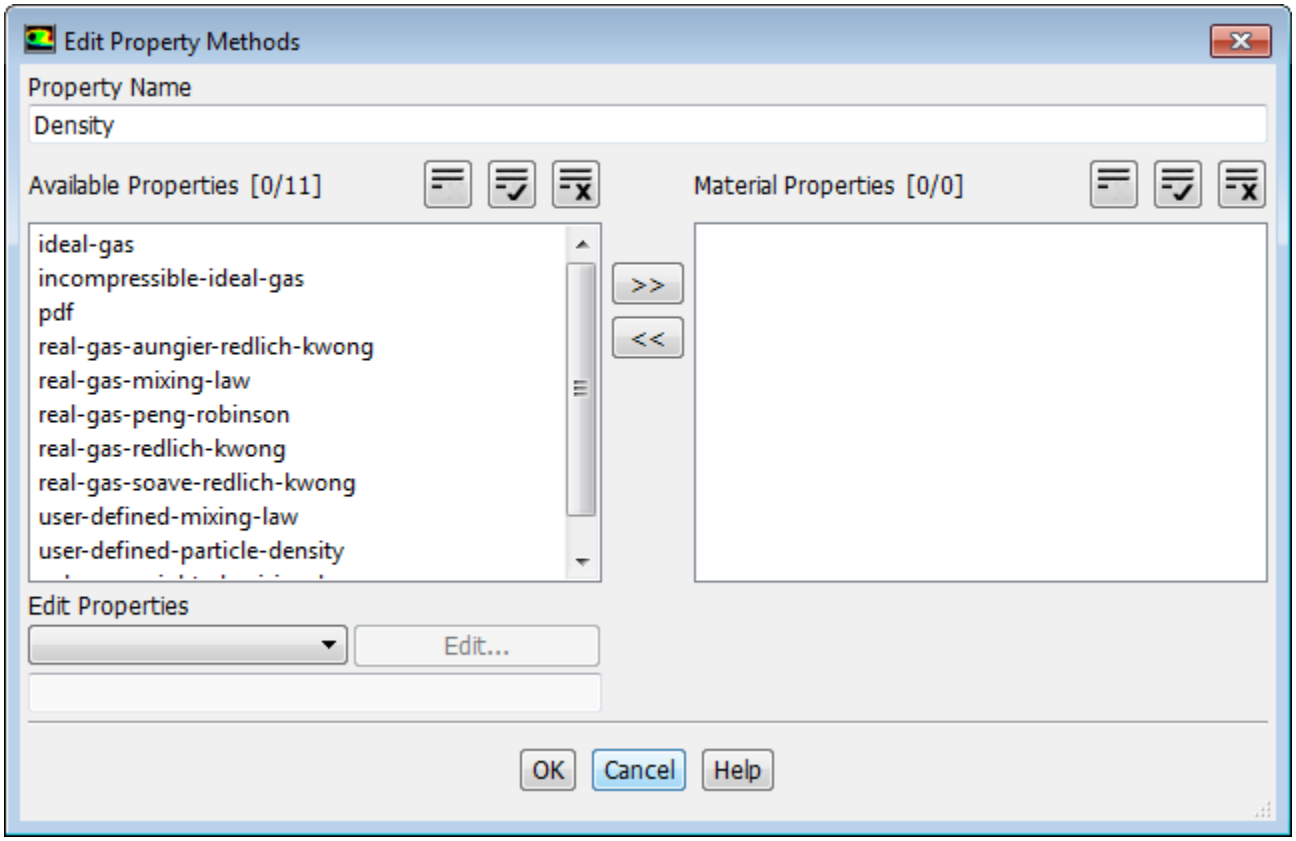
c. Depending on the type of material selected in the **Types** list, properties applicable to that material type will appear in the **Available Properties** list. Select the properties that are applicable for the material that you are defining by clicking on them.

d. Click the ≫ button to move these properties to the **Material Properties** list on the right and click **Apply**. You can use the ≪ button to move the property from the **Material Properties** list to the **Available Properties** list.

5. To edit the parameters that define a property, select the property in the **Material Properties** list and click **Edit...**. This opens the Edit Property Methods Dialog Box (p. 2316).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.10: Edit Property Methods Dialog Box



a. The methods that can be used to define the selected property are listed in the **Available Properties** list. You can select one or more methods and specify them for the material that you are defining, by selecting and moving them to the **Material Properties** list.

b. To modify each of these methods, you can select the method in the **Edit Properties** drop-down list and click **Edit...**. This will open the corresponding property dialog box, where you can modify the parameters used by the property method. Refer to Defining Properties Using Temperature-Dependent Functions (p. 475) to Real Gas Models (p. 532) for details of these properties, methods used to define the properties and the parameters for each method.

c. Click **OK** in the Edit Property Methods Dialog Box (p. 2316).

6. Click **Apply** in the Material Properties Dialog Box (p. 2315).

7. Click **Save** in the User-Defined Database Materials Dialog Box (p. 2313) to save the changes to the new materials database.

Similarly, you can also append new materials and click save to append these materials to the existing database.

7.1.4.7. Deleting Materials from a Database

To delete a material from a database, click the **User-Defined Database** button in the Open Database Dialog Box (p. 2313). Select the database and click **OK** in the Open Database Dialog Box (p. 2313). Select the **Material Type** and the materials that you want to delete in the **User-Defined Materials** list and click **Delete**. Click **Save** to save the database.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2. Defining Properties Using Temperature-Dependent Functions

Material properties can be defined as functions of temperature. For most properties, you can define a polynomial, piecewise-linear, or piecewise-polynomial function of temperature.

- polynomial:

$$\phi(T) = A_1 + A_2 T + A_3 T^2 + \ldots \tag{7.1}$$

- piecewise-linear:

$$\phi(T) = \phi_n + \frac{\phi_{n+1} - \phi_n}{T_{n+1} - T_n}(T - T_n) \tag{7.2}$$

  where $1 \le n \le N$ and N is the number of segments

- piecewise-polynomial:

$$\begin{aligned} &\text{for } T_{min,1} \le T < T_{max,1}: \phi(T) = A_1 + A_2 T + A_3 T^2 + \ldots \\ &\text{for } T_{min,2} \le T < T_{max,2}: \phi(T) = B_1 + B_2 T + B_3 T^2 + \ldots \end{aligned} \tag{7.3}$$

In the equations above, ϕ is the property.

Important

If you define a polynomial or piecewise-polynomial function of temperature, the temperature in the function is always in units of Kelvin or Rankine. If you use Celsius or Kelvin as the temperature unit, then polynomial coefficient values must be entered in terms of Kelvin. If you use Fahrenheit or Rankine as the temperature unit, enter the values in terms of Rankine.

Some properties have additional functions available and for some only a subset of these three functions can be used. See the section on the property in question to determine which temperature-dependent functions you can use.

For additional information, see the following sections:

7.2.1. Inputs for Polynomial Functions
7.2.2. Inputs for Piecewise-Linear Functions
7.2.3. Inputs for Piecewise-Polynomial Functions
7.2.4. Checking and Modifying Existing Profiles

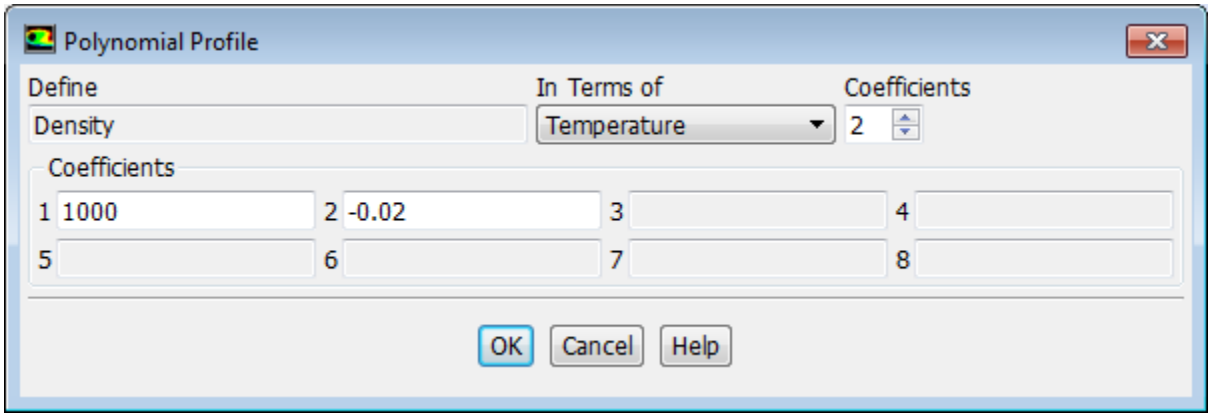
7.2.1. Inputs for Polynomial Functions

To define a polynomial function of temperature for a material property, do the following:

1. In the Create/Edit Materials Dialog Box (p. 2302), choose **polynomial** in the drop-down list to the right of the property name (for example, **Density**). The Polynomial Profile Dialog Box (p. 2318) (Figure 7.11: The Polynomial Profile Dialog Box (p. 476)) will open automatically.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.11: The Polynomial Profile Dialog Box



Since this is a modal dialog box, the solver will not allow you to do anything else until you perform the following steps.

a. Specify the number of **Coefficients** up to 8 coefficients are available. The number of coefficients defines the order of the polynomial. The default of 1 defines a polynomial of order 0. The property will be constant and equal to the single coefficient A_1. An input of 2 defines a polynomial of order 1 and the property will vary linearly with temperature and so on.

b. Define the coefficients. Coefficients **1**, **2**, **3**,... correspond to A_1, A_2, A_3,... in Equation 7.1 (p. 475). The dialog box in Figure 7.11: The Polynomial Profile Dialog Box (p. 476) shows the inputs for the following function:

$$\rho(T) = 1000 - 0.02T \tag{7.4}$$

Important

To ensure the robustness of simulations when using polynomials to specify material property, set the solution limits so that the values returned by the polynomial expression are physically valid. For example, when the density of a material is specified as a function of temperature using a polynomial expression, set the proper limits on the temperature range so that the density is always non-negative. For more information on setting solution limits see Setting Solution Limits (p. 1586).

Also, note the restriction on the units for temperature, as described in the previous section.

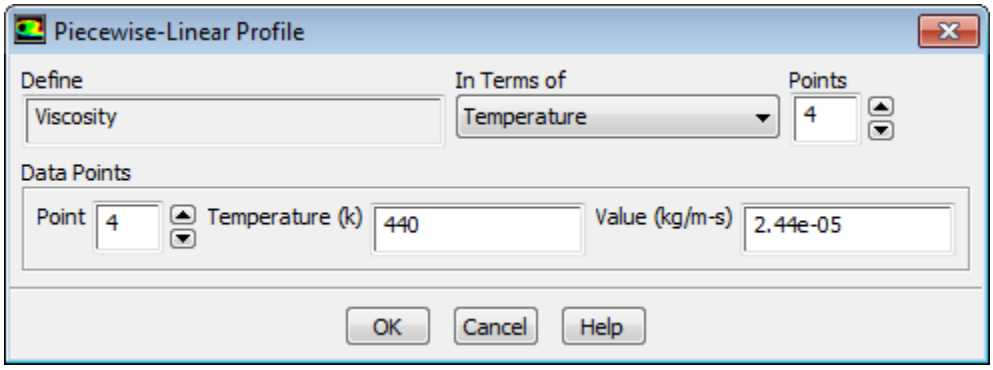
7.2.2. Inputs for Piecewise-Linear Functions

To define a piecewise-linear function of temperature for a material property, do the following:

1. In the Create/Edit Materials Dialog Box (p. 2302), choose **piecewise-linear** in the drop-down list to the right of the property name (for example, **Viscosity**). The Piecewise-Linear Profile Dialog Box (p. 2319) (Figure 7.12: The Piecewise-Linear Profile Dialog Box (p. 477)) will open automatically.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.12: The Piecewise-Linear Profile Dialog Box



Since this is a modal dialog box, the solver will not allow you to do anything else until you perform the following steps.

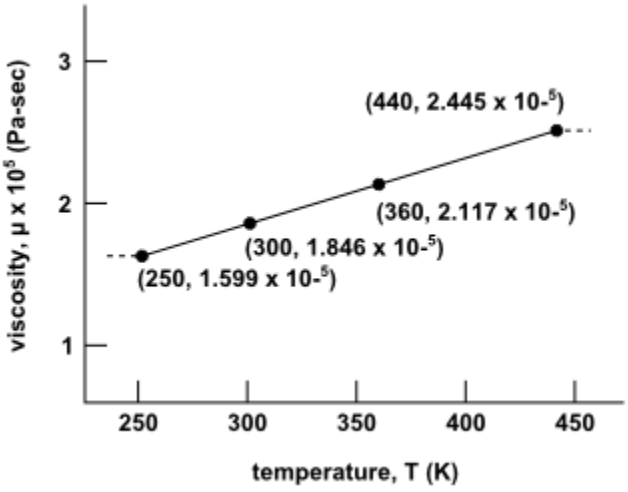
a. Set the number of **Points** defining the piecewise distribution.

b. Under **Data Points**, enter the data pairs for each point. First enter the independent and dependent variable values for **Point** 1, then increase the **Point** number and enter the appropriate values for each additional pair of variables. The pairs of points must be supplied in the order of increasing value of temperature. The solver will *not* sort them for you. A maximum of 50 piecewise points can be defined for each property. The dialog box in Figure 7.12: The Piecewise-Linear Profile Dialog Box (p. 477) shows the final inputs for the profile depicted in Figure 7.13: Piecewise-Linear Definition of Viscosity as a Function of Temperature (p. 477).

Important

If the temperature exceeds the maximum **Temperature** (T_{max}) you have specified for the profile, ANSYS Fluent will use the **Value** corresponding to T_{max}. If the temperature falls below the minimum **Temperature** (T_{min}) specified for your profile, ANSYS Fluent will use the **Value** corresponding to T_{min}.

Figure 7.13: Piecewise-Linear Definition of Viscosity as a Function of Temperature

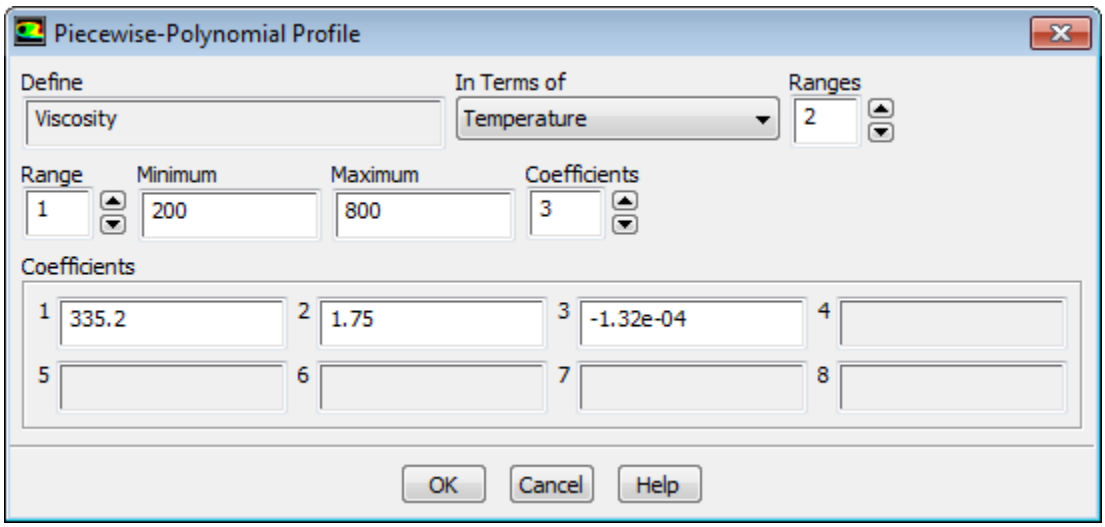


7.2.3. Inputs for Piecewise-Polynomial Functions

To define a piecewise-polynomial function of temperature for a material property, follow these steps:

1. In the Create/Edit Materials Dialog Box (p. 2302), choose **piecewise-polynomial** in the drop-down list to the right of the property name (for example, **Cp**). The Piecewise-Polynomial Profile Dialog Box (p. 2319) (Figure 7.14: The Piecewise-Polynomial Profile Dialog Box (p. 478)) will open automatically. Since this is a modal dialog box, first perform the following steps.

   Figure 7.14: The Piecewise-Polynomial Profile Dialog Box



2. Specify the number of **Ranges**. For the example of Equation 7.5 (p. 478), two ranges of temperatures are defined:

$$c_p(T) = \begin{cases} 429.929 + 1.784T - 1.966\times10^{-3}T^2 \\ \quad + 1.297\times10^{-6}T^3 - 4.000\times10^{-10}T^4 & for\ 300 \le T < 1000 \\ 841.377 + 0.593T - 2.415\times10^{-4}T^2 \\ \quad + 4.523\times10^{-8}T^3 - 3.153\times10^{-12}T^4 & for\ 1000 \le T < 5000 \end{cases} \tag{7.5}$$

   You may define up to three ranges. The ranges must be supplied in the order of increasing value of temperature. The solver will *not* sort them for you.

3. For the first range (**Range** = 1), specify the **Minimum** and **Maximum** temperatures, and the number of **Coefficients**. (Up to eight coefficients are available.) The number of coefficients defines the order of the polynomial. The default of 1 defines a polynomial of order 0. The property will be constant and equal to the single coefficient A_1. An input of 2 defines a polynomial of order 1. The property will vary linearly with temperature and so on.

4. Define the coefficients. Coefficients **1**, **2**, **3**,... correspond to A_1, A_2, A_3,... in Equation 7.3 (p. 475). The dialog box in Figure 7.14: The Piecewise-Polynomial Profile Dialog Box (p. 478) shows the inputs for the first range of Equation 7.5 (p. 478).

5. Increase the value of **Range** and enter the **Minimum** and **Maximum** temperatures, number of **Coefficients**, and the **Coefficients** (B_1, B_2, B_3,...) for the next range. Repeat if there is a third range.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.4. Checking and Modifying Existing Profiles

If you want to check or change the coefficients, data pairs, or ranges for a previously-defined profile, click the **Edit...** button to the right of the property name. The appropriate dialog box will open, and you can check or modify the inputs as desired.

Important

In the Fluent Database Materials Dialog Box (p. 2312), you cannot edit the profiles, but you can examine them by clicking on the **View...** button (instead of the **Edit...** button.)

7.3. Density

ANSYS Fluent provides several options for definition of the fluid density:

- constant density
- temperature and/or composition-dependent density
- pressure-dependent density

Each of these input options and the governing physical models are explained in the following sections. In all cases, you will define the **Density** in the Create/Edit Materials Dialog Box (p. 2302).

Setup → **Materials**

For additional information, see the following sections:
7.3.1. Defining Density for Various Flow Regimes
7.3.2. Input of Constant Density
7.3.3. Inputs for the Boussinesq Approximation
7.3.4. Compressible Liquid Density Method
7.3.5. Density as a Profile Function of Temperature
7.3.6. Incompressible Ideal Gas Law
7.3.7. Ideal Gas Law for Compressible Flows
7.3.8. Composition-Dependent Density for Multicomponent Mixtures

7.3.1. Defining Density for Various Flow Regimes

The selection of density in ANSYS Fluent is very important. Set the density relationship based on your flow regime.

- For compressible flows, the ideal gas law is the appropriate density relationship.
- For incompressible flows, you may choose one of the following methods:
  - Constant density, if you do not want density to be a function of temperature.
  - The incompressible ideal gas law, when pressure variations are small enough that the flow is fully incompressible but you want to use the ideal gas law to express the relationship between density and temperature (for example, for a natural convection problem).
  - Density as a polynomial, piecewise-linear, or piecewise-polynomial function of temperature, when the density is a function of temperature only, as in a natural convection problem.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– The Boussinesq model, for natural convection problems involving small changes in temperature.

– The compressible liquid density method allows you to model compressible liquids under high pressures.

7.3.1.1. Mixing Density Relationships in Multiple-Zone Models

If your model has multiple fluid zones that use different materials, you should be aware of the following:

- For calculations with the pressure-based solver that do not use one of the general multiphase models (Solution Strategies for Multiphase Modeling (p. 1504)), the compressible ideal gas law cannot be mixed with any other density methods. This means that if the compressible ideal gas law is used for one material, it must be used for all materials.

  This restriction does not apply to the density-based solvers.

- There is only one specified operating pressure and one specified operating temperature. This means that if you are using the ideal gas law for more than one material, they will share the same operating pressure. If you are using the Boussinesq model for more than one material, they will share the same operating temperature.

7.3.2. Input of Constant Density

If you want to define the density of the fluid as a constant, select **constant** in the **Density** drop-down list under **Properties** in the Create/Edit Materials Dialog Box (p. 2302). Enter the value of density for the material.

For the default fluid (air), the density is 1.225 kg/m^3.

7.3.3. Inputs for the Boussinesq Approximation

To enable the Boussinesq approximation for density, choose **boussinesq** from the **Density** drop-down list in the Create/Edit Materials Dialog Box (p. 2302) and specify a constant value for **Density**. You will also need to set the **Thermal Expansion Coefficient**, as well as relevant operating conditions, as described in The Boussinesq Model (p. 833).

7.3.4. Compressible Liquid Density Method

The compressible liquid treatment enables you to model liquid compressibility under high pressure applications. Fluent models compressible liquids using the Tait equation of state, which establishes a nonlinear relationship between density and pressure under isothermal conditions.

The compressible liquid treatment also helps in reducing unphysical pressure spikes that appear in moving and dynamic mesh applications, especially during solid-fluid interactions.

The Tait equation can be represented in terms of pressure and density using the following relationship:

$$p = a + b\rho^n \tag{7.6}$$

where, a and b are coefficients that can be determined by assuming that the bulk modulus is a linear function of pressure. The values of coefficients a and b are based on the reference state values of pressure, density, and bulk modulus.

The simplified form of the Tait equation can be written as:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\left(\frac{\rho}{\rho_0}\right)^n = \frac{K}{K_0} \tag{7.7}$$

where,

$$K = K_0 + n\Delta p \tag{7.8}$$

and

$$\Delta p = p - p_0 \tag{7.9}$$

where,

p_0 = Reference liquid pressure (Absolute)

ρ_0 = Reference liquid density (Density at reference pressure, p_0)

K_0 = Reference bulk modulus (Bulk modulus at reference pressure, p_0)

n = Density exponent

p = Liquid pressure (Absolute)

ρ = Liquid density at pressure, p

K = Bulk modulus at pressure, p

The density ratio is limited to the range:

$$\rho_{r,min} < \frac{\rho}{\rho_0} < \rho_{r,max} \tag{7.10}$$

where,

$\rho_{r,min}$= Minimum density ratio limit, when $p<p_0$

$\rho_{r,max}$= Maximum density ratio limit, when $p>p_0$

The speed of sound, c, is calculated as:

$$c = \sqrt{\frac{K}{\rho}} \tag{7.11}$$

Note

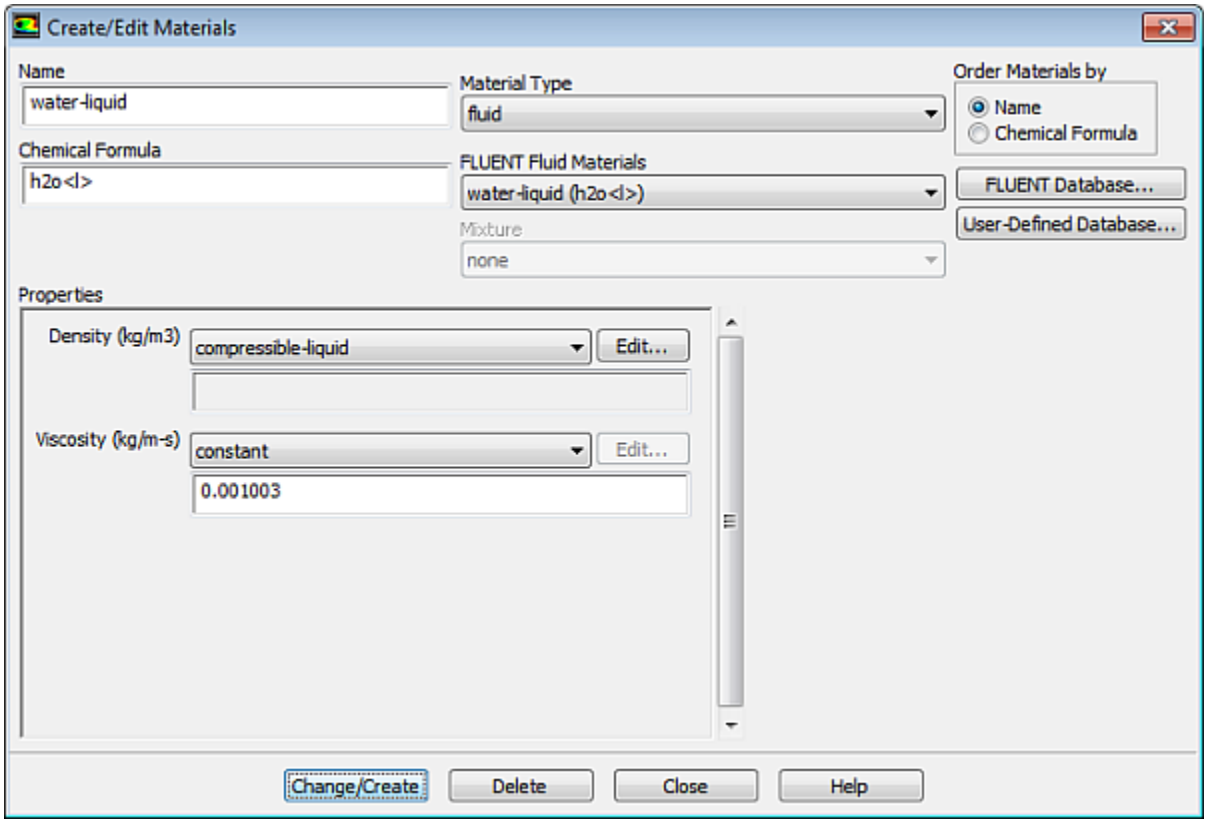
You can postprocess the speed of sound for compressible liquid materials. For multiphase models, sound-speed postprocessing is available at mixture and/or phase level, depending on the model selected. Sound-speed postprocessing is also enabled if you describe the speed of sound for a user-defined density.

7.3.4.1. Compressible Liquid Inputs

Select the `compressible-liquid` under **Density** in the **Create/Edit Materials** panel as seen in Figure 7.15: Compressible Liquid Materials Setting (p. 482).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.15: Compressible Liquid Materials Setting

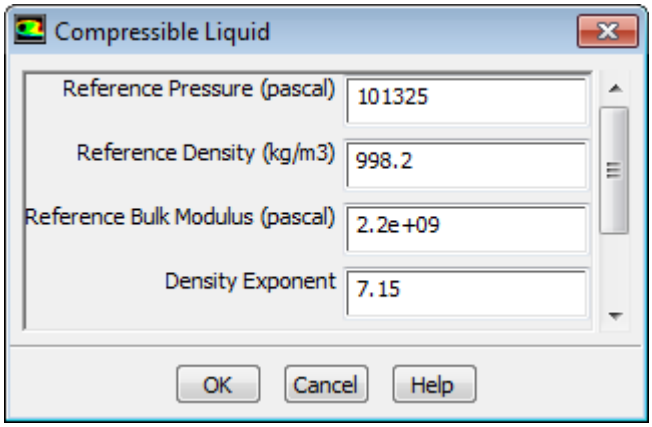


In the **Compressible Liquid** dialog box you need to specify values for the following settings:

- Reference Pressure
- Reference Density
- Reference Bulk Modulus
- Density Exponent
- Maximum Density Ratio Limit
- Minimum Density Ratio Limit

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.16: Compressible Liquid Density Settings Panel



Note

- ANSYS Fluent automatically fills user input values for certain materials based on open literature. If data is not available Fluent will fill in values of zero. Therefore, you must verify that suitable values are entered for your application.
- When Fluent automatically fills in values, the default value of the density exponent is set close to the value for water-liquid.
- The value for the density exponent is generally found by calibrating against data for density, pressure, bulk modulus, and speed of sound. If the value is unknown, it is recommended that you use a value of 1, which corresponds to a linear relationship between density and pressure.

You can set the compressible-liquid density method using the following text command:

`define` → `materials` → `change-create`

In the text command interface, set the `change Density?` option to `yes`, and set `new method [constant]` to `compressible-liquid` as shown below:

```
/define/materials> change-create

material-name> water-liquid
material name [water-liquid]
water-liquid is a fluid
change Density? [no] yes

Density
methods: (constant ideal-gas incompressible-ideal-gas real-gas-soave-redlich-kwong real-gas-peng-robinson
real-gas-aungier-redlich-kwong real-gas-redlich-kwong boussinesq piecewise-linear piecewise-polynomial
polynomial compressible-liquid user-defined)
new method [constant] compressible-liquid
Reference Pressure (pascal) [101325]
Reference Density (kg/m3) [998.2000000000001]
Reference Bulk Modulus (pascal) [2200000000]
Density Exponent [7.15]
Maximum Density Ratio Limit [1.1]
Minimum Density Ratio Limit [0.9]
```

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.3.4.2. Compressible Liquid Density Method Availability

The compressible liquid density method is available with fluid materials or with components of a mixture material having the density method set to `volume-weighted-mixing-law` for both single and multiphase cases.

This method is not available with the density-based solver.

7.3.5. Density as a Profile Function of Temperature

If you are modeling a problem that involves heat transfer, you can define the density as a function of temperature. Three types of functions are available:

- piecewise-linear:

$$\rho(T)=\rho_n+\frac{\rho_{n+1}-\rho_n}{T_{n+1}-T_n}(T-T_n) \tag{7.12}$$

- piecewise-polynomial:

$$\begin{aligned}&\text{for } T_{min,1}\le T<T_{max,1}: \rho(T)=A_1+A_2T+A_3T^2+\ldots\\&\text{for } T_{min,2}\le T<T_{max,2}: \rho(T)=B_1+B_2T+B_3T^2+\ldots\end{aligned} \tag{7.13}$$

- polynomial:

$$\rho(T)=A_1+A_2T+A_3T^2+\ldots \tag{7.14}$$

For one of the these methods, select **piecewise-linear**, **piecewise-polynomial**, or **polynomial** in the **Density** drop-down list. You can enter the data pairs (T_n,ρ_n), ranges and coefficients, or coefficients that describe these functions using the Create/Edit Materials Dialog Box (p. 2302), as described in Defining Properties Using Temperature-Dependent Functions (p. 475).

7.3.6. Incompressible Ideal Gas Law

In ANSYS Fluent, if you choose to define the density using the ideal gas law for an incompressible flow, the solver will compute the density as

$$\rho=\frac{p_{op}}{\frac{R}{M_w}T} \tag{7.15}$$

where,

R = the universal gas constant

M_w = the molecular weight of the gas

p_{op} = the operating pressure

In this form, the density depends only on the operating pressure and not on the local relative pressure field.

7.3.6.1. Density Inputs for the Incompressible Ideal Gas Law

The inputs for the incompressible ideal gas law are as follows:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Enable the ideal gas law for an incompressible fluid by choosing **incompressible-ideal-gas** from the drop-down list to the right of **Density** in the Create/Edit Materials Dialog Box (p. 2302).

   Specify the incompressible ideal gas law individually for each material that you want to use it for. See Composition-Dependent Density for Multicomponent Mixtures (p. 486) for information on specifying the incompressible ideal gas law for mixtures.

2. Set the operating pressure by defining the **Operating Pressure** in the Operating Conditions Dialog Box (p. 2382).

   Setup → **Cell Zone Conditions** → **Operating Conditions...**

   Important

   By default, operating pressure is set to 101325 Pa. The input of the operating pressure is of great importance when you are computing density with the ideal gas law. See Operating Pressure (p. 530) for recommendations on setting appropriate values for the operating pressure.

3. Set the molecular weight of the homogeneous or single-component fluid (if no chemical species transport equations are to be solved), or the molecular weights of each fluid material (species) in a multicomponent mixture. For each fluid material, enter the value of the **Molecular Weight** in the Create/Edit Materials Dialog Box (p. 2302).

7.3.7. Ideal Gas Law for Compressible Flows

For compressible flows, the gas law is as following:

$$\rho = \frac{p_{op} + p}{\frac{R}{M_w} T} \tag{7.16}$$

where,

p = the local relative (or gauge) pressure predicted by ANSYS Fluent

p_{op} = the operating pressure

7.3.7.1. Density Inputs for the Ideal Gas Law for Compressible Flows

The inputs for the ideal gas law are as follows:

1. Enable the ideal gas law for a compressible fluid by choosing **ideal-gas** from the drop-down list to the right of **Density** in the Create/Edit Materials Dialog Box (p. 2302).

   Specify the ideal gas law individually for each material that you want to use it for. See Composition-Dependent Density for Multicomponent Mixtures (p. 486) for information on specifying the ideal gas law for mixtures.

2. Set the operating pressure by defining the **Operating Pressure** in the Operating Conditions Dialog Box (p. 2382).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → Cell Zone Conditions → Operating Conditions...

Important

The input of the operating pressure is of great importance when you are computing density with the ideal gas law. Equation 7.16 (p. 485) notes that the operating pressure is added to the relative pressure field computed by the solver, yielding the absolute static pressure. See Operating Pressure (p. 530) for recommendations on setting appropriate values for the operating pressure. By default, operating pressure is set to 101325 Pa.

3. Set the molecular weight of the homogeneous or single-component fluid (if no chemical species transport equations are to be solved), or the molecular weights of each fluid material (species) in a multicomponent mixture. For each fluid material, enter the value of the **Molecular Weight** in the Create/Edit Materials Dialog Box (p. 2302).

7.3.8. Composition-Dependent Density for Multicomponent Mixtures

If you are solving species transport equations, set properties for the mixture material and for the constituent fluids (species), as described in detail in Defining Properties for the Mixture and Its Constituent Species (p. 972). To define a composition-dependent density for a mixture, do the following:

1. Select the density method:

   - For non-ideal-gas mixtures, select the **volume-weighted-mixing-law** method for the mixture material in the drop-down list to the right of **Density** in the Create/Edit Materials Dialog Box (p. 2302).
   - If you are modeling compressible flow, select **ideal-gas** for the mixture material in the drop-down list to the right of **Density** in the Create/Edit Materials Dialog Box (p. 2302).
   - If you are modeling incompressible flow using the ideal gas law, select **incompressible-ideal-gas** for the mixture material in the **Density** drop-down list in the Create/Edit Materials Dialog Box (p. 2302).
   - If you have a user-defined function that you want to use to model the density, you can choose either the **user-defined** method or the **user-defined-mixing-law** method for the mixture material in the drop-down list.

   The only difference between the **user-defined-mixing-law** and the **user-defined** option for specifying density, viscosity and thermal conductivity of mixture materials, is that with the **user-defined-mixing-law** option, the individual properties of the species materials can also be specified. (Note that only the constant, the polynomial methods and the user-defined methods are available.)

2. Click **Change/Create**.

3. If you have selected **volume-weighted-mixing-law**, define the density for each of the fluid materials that comprise the mixture. You may define constant or compressible-liquid or (if applicable) temperature-dependent densities for the individual species.

4. If you selected **user-defined-mixing-law**, define the density for each of the fluid materials that comprise the mixture. You may define constant, or (if applicable) temperature-dependent densities, or user-defined densities for the individual species. More information on defining properties with user-defined functions can be found in the Fluent Customization Manual.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you are modeling a non-ideal-gas mixture, ANSYS Fluent will compute the mixture density as

$$\rho = \frac{1}{\sum_i \frac{Y_i}{\rho_i}} \tag{7.17}$$

where Y_i is the mass fraction and ρ_i is the density of species i.

For compressible flows, the gas law has the following form:

$$\rho = \frac{p_{op} + p}{RT \sum_i \frac{Y_i}{M_{w,i}}} \tag{7.18}$$

where,

p = the local relative (or gauge) pressure predicted by ANSYS Fluent

R = the universal gas constant

Y_i = the mass fraction of species i

$M_{w,i}$ = the molecular weight of species i

p_{op} = the operating pressure

In ANSYS Fluent, if you choose to define the density using the ideal gas law for an incompressible flow, the solver will compute the density as

$$\rho = \frac{p_{op}}{RT \sum_i \frac{Y_i}{M_{w,i}}} \tag{7.19}$$

where,

R = the universal gas constant

Y_i = the mass fraction of species i

$M_{w,i}$ = the molecular weight of species i

p_{op} = the operating pressure

7.4. Viscosity

ANSYS Fluent provides several options for definition of the fluid viscosity:

- constant viscosity
- temperature-dependent and/or composition-dependent viscosity
- kinetic theory
- non-Newtonian viscosity
- user-defined function

Each of these input options and the governing physical models are detailed in this section. (User-defined functions are described in the Fluent Customization Manual). In all cases, define the **Viscosity** in the Create/Edit Materials Dialog Box (p. 2302).

Setup → Materials

Viscosities are input as dynamic viscosity (μ) in units of kg/m-s in SI units or lb_m/ft-s in British units. ANSYS Fluent does not ask for input of the kinematic viscosity (ν).

For additional information, see the following sections:

7.4.1. Input of Constant Viscosity
7.4.2. Viscosity as a Function of Temperature
7.4.3. Defining the Viscosity Using Kinetic Theory
7.4.4. Composition-Dependent Viscosity for Multicomponent Mixtures
7.4.5. Viscosity for Non-Newtonian Fluids

7.4.1. Input of Constant Viscosity

If you want to define the viscosity of your fluid as a constant, select **constant** in the **Viscosity** drop-down list in the Create/Edit Materials Dialog Box (p. 2302), and enter the value of viscosity for the fluid.

For the default fluid (air), the viscosity is 1.7894×10^{-5} kg/m-s.

7.4.2. Viscosity as a Function of Temperature

If you are modeling a problem that involves heat transfer, you can define the viscosity as a function of temperature. Five types of functions are available.

- piecewise-linear:

$$\mu(T)=\mu_n+\frac{\mu_{n+1}-\mu_n}{T_{n+1}-T_n}(T-T_n) \tag{7.20}$$

- piecewise-polynomial:

$$\begin{aligned}&\text{for } T_{min,1}\le T<T_{max,1}: \mu(T)=A_1+A_2T+A_3T^2+\ldots\\&\text{for } T_{min,2}\le T<T_{max,2}: \mu(T)=B_1+B_2T+B_3T^2+\ldots\end{aligned} \tag{7.21}$$

- polynomial:

$$\mu(T)=A_1+A_2T+A_3T^2+\ldots \tag{7.22}$$

- Sutherland's law
- power law

Important

The power law described here is different from the non-Newtonian power law described in Viscosity for Non-Newtonian Fluids (p. 492).

For one of the first three, select **piecewise-linear**, **piecewise-polynomial**, **polynomial** in the **Viscosity** drop-down list and then enter the data pairs (T_n,μ_n), ranges and coefficients, or coefficients that describe these functions Defining Properties Using Temperature-Dependent Functions (p. 475). For Sutherland's law or the power law, choose **sutherland** or **power-law** respectively in the drop-down list and enter the parameters.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.2.1. Sutherland Viscosity Law

Sutherland's viscosity law resulted from a kinetic theory by Sutherland (1893) using an idealized intermolecular-force potential. The formula is specified using two or three coefficients.

Sutherland's law with two coefficients has the form

$$\mu=\frac{C_1 T^{3/2}}{T+C_2} \tag{7.23}$$

where,

μ = the viscosity in kg/m-s

T = the static temperature in K

C_1 and C_2 = the coefficients

For air at moderate temperatures and pressures, $C_1=1.458\times10^{-6}$ kg/m-s-$K^{1/2}$, and $C_2=110.4$ K.

Sutherland's law with three coefficients has the form

$$\mu=\mu_0\left(\frac{T}{T_0}\right)^{3/2}\frac{T_0+S}{T+S} \tag{7.24}$$

where,

μ = the viscosity in kg/m-s

T = the static temperature in K

μ_0 = reference value in kg/m-s

T_0 = reference temperature in K

S = an effective temperature in K (Sutherland constant)

For air at moderate temperatures and pressures, $\mu_0=1.716\times10^{-5}$ kg/m-s, T_0 = 273.11 K, and S = 110.56 K.

7.4.2.1.1. Inputs for Sutherland's Law

To use Sutherland's law, choose **sutherland** in the drop-down list to the right of **Viscosity**. The Sutherland Law Dialog Box (p. 2322) will open, and you can enter the coefficients as follows:

1. Select the **Two Coefficient Method** or the **Three Coefficient Method**.

   Important

   Use SI units if you choose the two-coefficient method.

2. For the **Two Coefficient Method**, set **C1** and **C2**. For the **Three Coefficient Method**, set the **Reference Viscosity** μ_0, the **Reference Temperature** T_0, and the **Effective Temperature** S.

7.4.2.2. Power-Law Viscosity Law

Another common approximation for the viscosity of dilute gases is the power-law form. For dilute gases at moderate temperatures, this form is considered to be slightly less accurate than Sutherland's law.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A power-law viscosity law with two coefficients has the form

$$\mu = BT^n \tag{7.25}$$

where,

μ = the viscosity in kg/m-s

T = the static temperature in K

B = a dimensional coefficient

For air at moderate temperatures and pressures, $B = 4.093 \times 10^{-7}$, and $n = 2/3$.

A power-law viscosity law with three coefficients has the form

$$\mu = \mu_0 \left(\frac{T}{T_0} \right)^n \tag{7.26}$$

where,

μ = the viscosity in kg/m-s

T = the static temperature in K

T_0 = a reference value in K

μ_0 = a reference value in kg/m-s

For air at moderate temperatures and pressures, $\mu_0 = 1.716 \times 10^{-5}$ kg/m-s, $T_0 = 273$ K, and $n = 2/3$.

Important

The non-Newtonian power law for viscosity is described in Viscosity for Non-Newtonian Fluids (p. 492).

7.4.2.2.1. Inputs for the Power Law

To use the power law, choose **power-law** in the drop-down list to the right of **Viscosity**. The Power Law Dialog Box (p. 2323) will open, and you can enter the coefficients as follows:

1. Select the **Two Coefficient Method** or the **Three Coefficient Method**.

   Important

   Note that you must use SI units if you choose the two-coefficient method.

2. For the **Two Coefficient Method**, set **B** and the **Temperature Exponent** n. For the **Three Coefficient Method**, set the **Reference Viscosity** μ_0, the **Reference Temperature** T_0, and the **Temperature Exponent** n.

7.4.3. Defining the Viscosity Using Kinetic Theory

If you are using the ideal gas law (as described in Density (p. 479)), you have the option to define the fluid viscosity using kinetic theory as

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\mu = 2.67 \times 10^{-6} \frac{\sqrt{M_w T}}{\sigma^2 \Omega_\mu} \tag{7.27}$$

where,

μ is in units of kg/m-s,

T is in units of Kelvin

σ is in units of Angstroms

$\Omega_\mu = \Omega_\mu(T^*)$

M_w is the molecular weight

where

$$T^* = \frac{T}{(\varepsilon / k_B)} \tag{7.28}$$

The Lennard-Jones parameters, σ and ε / k_B, are inputs to the kinetic theory calculation that you supply by selecting **kinetic-theory** from the drop-down list to the right of **Viscosity** in the Create/Edit Materials Dialog Box (p. 2302). The solver will use these kinetic theory inputs in Equation 7.27 (p. 491) to compute the fluid viscosity. See Kinetic Theory Parameters (p. 529) for details about these inputs.

7.4.4. Composition-Dependent Viscosity for Multicomponent Mixtures

If you are modeling a flow that includes more than one chemical species (multicomponent flow), you have the option to define a composition-dependent viscosity. (Note that you can also define the viscosity of the mixture as a constant value or a function of temperature.)

To define a composition-dependent viscosity for a mixture, follow these steps:

1. For the mixture material, choose **mass-weighted-mixing-law** or, if you are using the ideal gas law for density, **ideal-gas-mixing-law** in the drop-down list to the right of **Viscosity**. If you have a user-defined function that you want to use to model the viscosity, you can choose either the **user-defined** method or the **user-defined-mixing-law** method for the mixture material in the drop-down list.

2. Click **Change/Create**.

3. Define the viscosity for each of the fluid materials that comprise the mixture. You may define constant or (if applicable) temperature-dependent viscosities for the individual species. You may also use kinetic theory for the individual viscosities, or specify a non-Newtonian viscosity, if applicable.

4. If you selected **user-defined-mixing-law**, define the viscosity for each of the fluid materials that comprise the mixture. You may define constant, or (if applicable) temperature-dependent viscosities, or user-defined viscosities for the individual species. More information on defining properties with user-defined functions can be found in the Fluent Customization Manual.

   The only difference between the **user-defined-mixing-law** and the **user-defined** option for specifying density, viscosity and thermal conductivity of mixture materials, is that with the **user-defined-mixing-law** option, the individual properties of the species materials can also be specified. (Note that only the constant, the polynomial methods and the user-defined methods are available.)

If you are using the ideal gas law, the solver will compute the mixture viscosity based on kinetic theory as

$$\mu = \sum_i \frac{X_i \mu_i}{\sum_j X_j \phi_{ij}} \tag{7.29}$$

where

$$\phi_{ij} = \frac{\left[1 + \left(\frac{\mu_i}{\mu_j}\right)^{1/2} \left(\frac{M_{w,j}}{M_{w,i}}\right)^{1/4}\right]^2}{\left[8\left(1 + \frac{M_{w,i}}{M_{w,j}}\right)\right]^{1/2}} \tag{7.30}$$

and X_i is the mole fraction of species i.

For non-ideal gas mixtures, the mixture viscosity is computed based on a simple mass fraction average of the pure species viscosities:

$$\mu = \sum_i Y_i \mu_i \tag{7.31}$$

7.4.5. Viscosity for Non-Newtonian Fluids

For incompressible Newtonian fluids, the shear stress is proportional to the rate-of-deformation tensor $\overline{\overline{D}}$:

$$\overline{\overline{\tau}} = \mu \overline{\overline{D}} \tag{7.32}$$

where $\overline{\overline{D}}$ is defined by

$$\overline{\overline{D}} = \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j}\right) \tag{7.33}$$

and μ is the viscosity, which is independent of $\overline{\overline{D}}$.

For some non-Newtonian fluids, the shear stress can similarly be written in terms of a non-Newtonian viscosity η:

$$\overline{\overline{\tau}} = \eta\left(\overline{\overline{D}}\right) \overline{\overline{D}} \tag{7.34}$$

In general, η is a function of all three invariants of the rate-of-deformation tensor $\overline{\overline{D}}$. However, in the non-Newtonian models available in ANSYS Fluent, η is considered to be a function of the shear rate $\dot{\gamma}$ only. $\dot{\gamma}$ is related to the second invariant of $\overline{\overline{D}}$ and is defined as

$$\dot{\gamma} = \sqrt{\frac{1}{2} \overline{\overline{D}} : \overline{\overline{D}}} \tag{7.35}$$

7.4.5.1. Temperature Dependent Viscosity

If the flow is non-isothermal, then the temperature dependence on the viscosity can be included along with the shear rate dependence. In this case, the total viscosity consists of two parts and is calculated as

$$\mu = \eta\left(\dot{\gamma}\right) H\left(T\right) \tag{7.36}$$

where H(T) is the temperature dependence, known as the Arrhenius law.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$H(T)=exp\left[\alpha\left(\frac{1}{T-T_0}-\frac{1}{T_\alpha-T_0}\right)\right] \tag{7.37}$$

where α is the ratio of the activation energy to the thermodynamic constant and T_α is a reference temperature for which H(T) = 1. T_0, which is the temperature shift, is set to 0 by default, and corresponds to the lowest temperature that is thermodynamically acceptable. Therefore T_α and T_0 are absolute temperatures. Temperature dependence is only included when the energy equation is enabled. Set the parameter α to 0 when you want temperature dependence to be ignored, even when the energy equation is solved.

ANSYS Fluent provides four options for modeling non-Newtonian flows:

- power law
- Carreau model for pseudo-plastics
- Cross model
- Herschel-Bulkley model for Bingham plastics

Important

- Note that the models listed above are not available when modeling turbulent flow.
- Note that the non-Newtonian power law described below is different from the power law described in Power-Law Viscosity Law (p. 489).
- Non-Newtonian model based on single fluid formulation is available for the mixture model and it is recommended that this should be attached to the primary phase.

Appropriate values for the input parameters for these models can be found in the literature (for example, [119] (p. 2893)).

7.4.5.2. Power Law for Non-Newtonian Viscosity

If you choose **non-newtonian-power-law** in the drop-down list to the right of **Viscosity**, non-Newtonian flow will be modeled according to the following power law for the non-Newtonian viscosity:

$$\eta=k\dot{\gamma}^{n-1}H(T) \tag{7.38}$$

where k and n are input parameters. k is a measure of the average viscosity of the fluid (the consistency index); n is a measure of the deviation of the fluid from Newtonian (the power-law index). The value of n determines the class of the fluid:

$n=1 \rightarrow$ Newtonian fluid

$n>1 \rightarrow$ shear-thickening (dilatant fluids)

$n<1 \rightarrow$ shear-thinning (pseudo-plastics)

7.4.5.2.1. Inputs for the Non-Newtonian Power Law

To use the non-Newtonian power law, choose **non-newtonian-power-law** in the drop-down list to the right of **Viscosity**. The Non-Newtonian Power Law Dialog Box (p. 2324) will open, and you can choose

between **Shear Rate Dependent** and **Shear Rate and Temperature Dependent**. Enter the **Consistency Index** k, **Power-Law Index** n, **Minimum** and **Maximum Viscosity Limit**, **Reference Temperature** T_{alpha}, and **Activation Energy/R**, α, which is the ratio of the activation energy to the thermodynamic constant.

7.4.5.3. The Carreau Model for Pseudo-Plastics

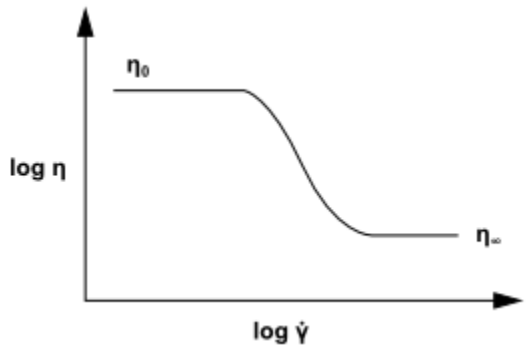
The power law model described in Equation 7.38 (p. 493) results in a fluid viscosity that varies with shear rate. For $\dot{\gamma} \to 0$, $\eta \to \eta_0$, and for $\dot{\gamma} \to \infty$, $\eta \to \eta_\infty$, where η_0 and η_∞ are, respectively, the upper and lower limiting values of the fluid viscosity.

The Carreau model attempts to describe a wide range of fluids by the establishment of a curve-fit to piece together functions for both Newtonian and shear-thinning ($n < 1$) non-Newtonian laws. In the Carreau model, the viscosity is

$$\eta = H(T)\left(\eta_\infty + \left(\eta_0 - \eta_\infty\right)\left[1 + \gamma^2\lambda^2\right]^{(n-1)/2}\right) \tag{7.39}$$

and the parameters n, λ, T_α, η_0, and η_∞ are dependent upon the fluid. λ is the time constant, n is the power-law index (as described above for the non-Newtonian power law), η_0 and η_∞ are, respectively, the zero- and infinite-shear viscosities, T_α is the reference temperature, and α is the ratio of the activation energy to thermodynamic constant. Figure 7.17: Variation of Viscosity with Shear Rate According to the Carreau Model (p. 494) shows how viscosity is limited by η_0 and η_∞ at low and high shear rates.

Figure 7.17: Variation of Viscosity with Shear Rate According to the Carreau Model

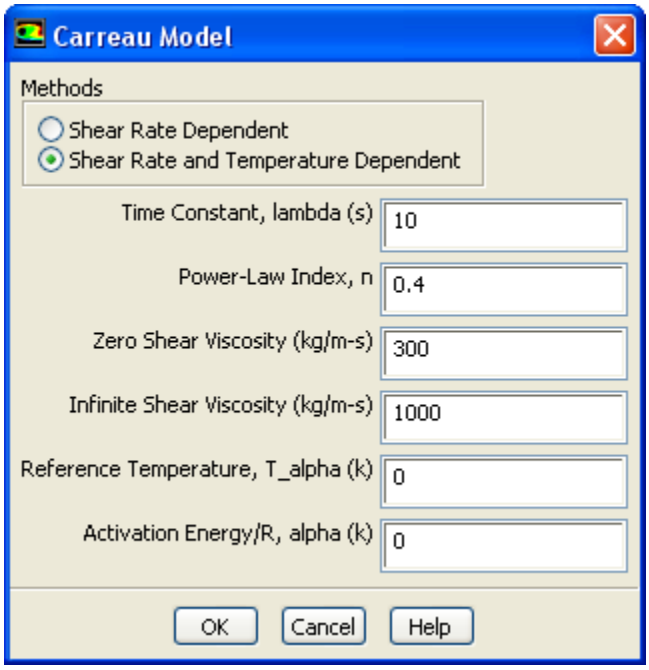


7.4.5.3.1. Inputs for the Carreau Model

To use the Carreau model, choose **carreau** in the drop-down list to the right of **Viscosity**. The Carreau Model Dialog Box (p. 2324) will open, and you can choose between **Shear Rate Dependent** and **Shear Rate and Temperature Dependent**. Enter the **Time Constant** λ, **Power-Law Index** n, **Reference Temperature** T_α, **Zero Shear Viscosity** η_0, **Infinite Shear Viscosity** η_∞, and **Activation Energy/R** α.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.18: The Carreau Model Dialog Box



7.4.5.4. Cross Model

The Cross model for viscosity is:

$$\eta = H(T)\frac{\eta_0}{1+(\lambda\dot{\gamma})^{1-n}} \tag{7.40}$$

where,

η_0 = zero-shear-rate viscosity

λ = natural time (that is, inverse of the shear rate at which the fluid changes from Newtonian to power-law behavior)

n = power-law index

The Cross model is commonly used to describe the low-shear-rate behavior of the viscosity.

7.4.5.4.1. Inputs for the Cross Model

To use the Cross model, choose **cross** in the drop-down list to the right of **Viscosity**. The Cross Model Dialog Box (p. 2325) will open, and you can choose between **Shear Rate Dependent** and **Shear Rate and Temperature Dependent**. Enter the **Zero Shear Viscosity** η_0, **Time Constant** λ, **Power-Law Index** n, **Reference Temperature** T_α, and **Activation Energy/R**, α, which is the ratio of the activation energy to the thermodynamic constant.

7.4.5.5. Herschel-Bulkley Model for Bingham Plastics

The power law model described above is valid for fluids for which the shear stress is zero when the strain rate is zero. Bingham plastics are characterized by a nonzero shear stress when the strain rate is zero.

$$\bar{\bar{\tau}} = \bar{\bar{\tau}}_0 + \eta\bar{\bar{D}} \tag{7.41}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where τ_0 is the yield stress:

- For $\tau<\tau_0$, the material remains rigid.
- For $\tau>\tau_0$, the material flows as a power-law fluid.

The Herschel-Bulkley model combines the effects of Bingham and power-law behavior in a fluid. For low strain rates ($\dot{\gamma}<\tau_0/\mu_0$), the "rigid" material acts like a very viscous fluid with viscosity μ_0. As the strain rate increases and the yield stress threshold, τ_0, is passed, the fluid behavior is described by a power law.

For $\dot{\gamma}>\dot{\gamma}_c$

$$\eta=\frac{\tau_0}{\dot{\gamma}}+k\dot{\gamma}^{n-1} \tag{7.42}$$

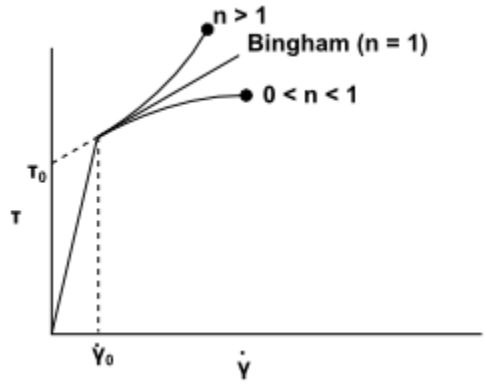
For $\dot{\gamma}<\dot{\gamma}_c$

$$\eta=\frac{\tau_0\left(2-\frac{\dot{\gamma}}{\dot{\gamma}_c}\right)}{\dot{\gamma}_c}+k\left(\dot{\gamma}_c\right)^{n-1}\left[(2-n)+(n-1)\frac{\dot{\gamma}}{\dot{\gamma}_c}\right] \tag{7.43}$$

where k is the consistency indexmetika, and n is the power-law index.

Figure 7.19: Variation of Shear Stress with Shear Rate According to the Herschel-Bulkley Model (p. 496) shows how shear stress (τ) varies with shear rate ($\dot{\gamma}$) for the Herschel-Bulkley model.

Figure 7.19: Variation of Shear Stress with Shear Rate According to the Herschel-Bulkley Model



If you choose the Herschel-Bulkley model for Bingham plastics, Equation 7.42 (p. 496) will be used to determine the fluid viscosity.

The Herschel-Bulkley model is commonly used to describe materials such as concrete, mud, dough, and toothpaste, for which a constant viscosity after a critical shear stress is a reasonable assumption. In addition to the transition behavior between a flow and no-flow regime, the Herschel-Bulkley model can also exhibit a shear-thinning or shear-thickening behavior depending on the value of n.

7.4.5.5.1. Inputs for the Herschel-Bulkley Model

To use the Herschel-Bulkley model, choose **herschel-bulkley** in the drop-down list to the right of **Viscosity**. The Herschel-Bulkley Dialog Box (p. 2326) will open, and you can choose between **Shear Rate Dependent** and **Shear Rate and Temperature Dependent**. Enter the **Consistency Index** k, **Power-**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Law Index n, **Yield Stress Threshold** τ_0, **Critical Shear Rate** $\dot{\gamma}_c$, **Reference Temperature** T_α, and the ratio of the activation energy to thermodynamic constant α, **Activation Energy/R**.

7.5. Thermal Conductivity

The thermal conductivity must be defined when heat transfer is active. You must define thermal conductivity when you are modeling energy and viscous flow.

ANSYS Fluent provides several options for definition of the thermal conductivity:

- constant thermal conductivity
- temperature- and/or composition-dependent thermal conductivity
- kinetic theory
- anisotropic (anisotropic, biaxial, orthotropic, cylindrical orthotropic, principal axes and principal values, user-defined anisotropic) (for solid materials only)
- user-defined

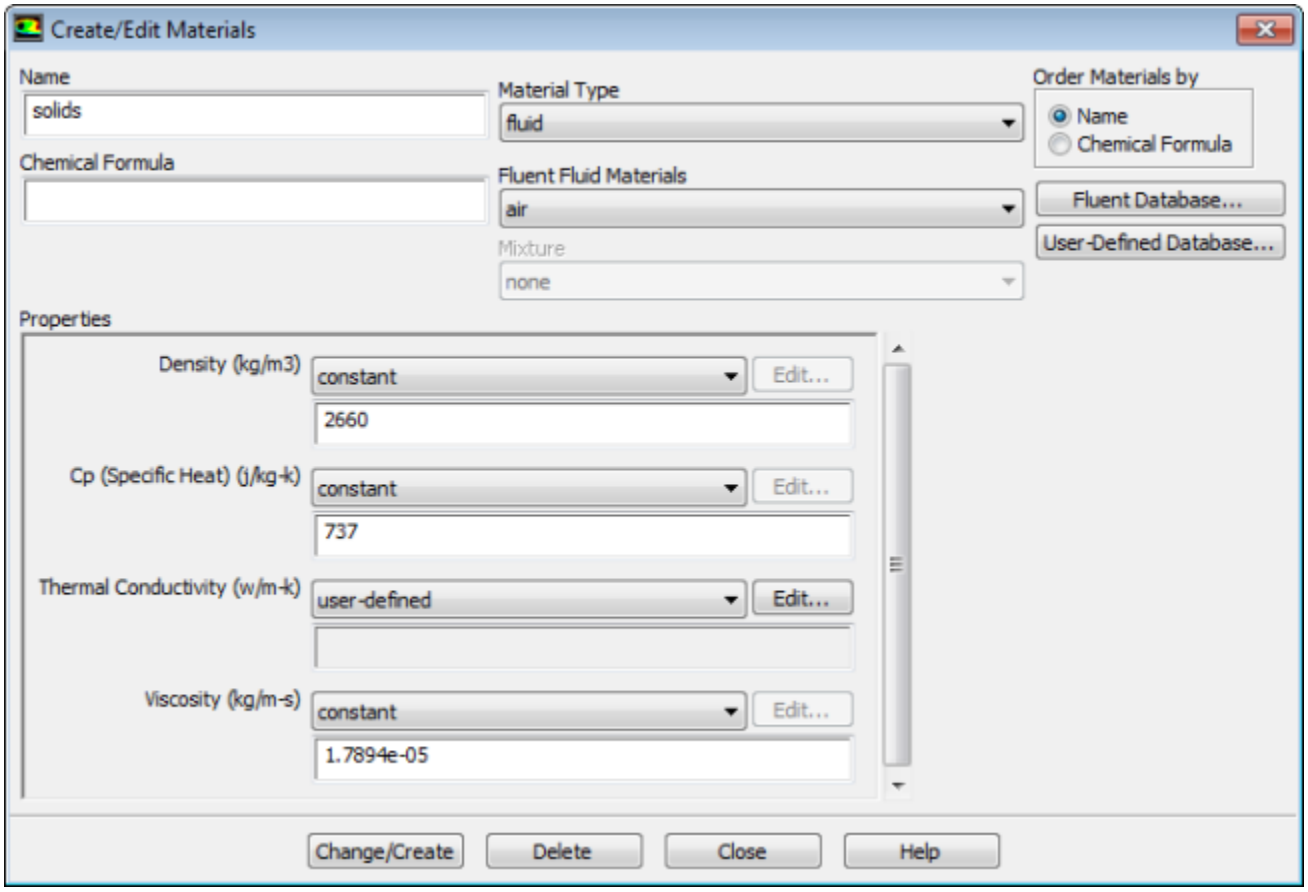
Each of these input options and the governing physical models are detailed in this section. User-defined functions (UDFs) are described in the Fluent Customization Manual.

In all cases, you will define the **Thermal Conductivity** in the Create/Edit Materials Dialog Box (p. 2302) (Figure 7.20: The Create/Edit Materials Dialog Box (p. 498)).

Setup → **Materials** → **Create/Edit...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.20: The Create/Edit Materials Dialog Box



Thermal conductivity is defined in units of W/m-K in SI units or BTU/hr-ft-°R in British units.

For additional information, see the following sections:

7.5.1. Constant Thermal Conductivity
7.5.2. Thermal Conductivity as a Function of Temperature
7.5.3. Thermal Conductivity Using Kinetic Theory
7.5.4. Composition-Dependent Thermal Conductivity for Multicomponent Mixtures
7.5.5. Anisotropic Thermal Conductivity for Solids

7.5.1. Constant Thermal Conductivity

If you want to define the thermal conductivity as a constant, check that **constant** is selected in the drop-down list to the right of **Thermal Conductivity** in the Create/Edit Materials Dialog Box (p. 2302) (Figure 7.20: The Create/Edit Materials Dialog Box (p. 498)), and enter the value of thermal conductivity for the material.

For the default fluid (air), the thermal conductivity is 0.0242 W/m-K.

7.5.2. Thermal Conductivity as a Function of Temperature

You can also choose to define the thermal conductivity as a function of temperature. Three types of functions are available:

- piecewise-linear:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$k(T) = k_n + \frac{k_{n+1} - k_n}{T_{n+1} - T_n}(T - T_n) \tag{7.44}$$

- piecewise-polynomial:

$$\begin{aligned} &\text{for } T_{min,1} \le T < T_{max,1}: k(T) = A_1 + A_2 T + A_3 T^2 + \ldots \\ &\text{for } T_{min,2} \le T < T_{max,2}: k(T) = B_1 + B_2 T + B_3 T^2 + \ldots \end{aligned} \tag{7.45}$$

- polynomial:

$$k(T) = A_1 + A_2 T + A_3 T^2 + \ldots \tag{7.46}$$

You can input the data pairs (T_n, k_n), ranges and coefficients A_i and B_i, or coefficients A_i that describe these functions using the Create/Edit Materials Dialog Box (p. 2302), as described in Defining Properties Using Temperature-Dependent Functions (p. 475).

7.5.3. Thermal Conductivity Using Kinetic Theory

If you are using the ideal gas law (as described in Density (p. 479)), you have the option to define the thermal conductivity using kinetic theory as

$$k = \frac{15}{4}\frac{R}{M_w}\mu\left[\frac{4}{15}\frac{c_p M_w}{R} + \frac{1}{3}\right] \tag{7.47}$$

where R is the universal gas constant, M_w is the molecular weight, μ is the material's specified or computed viscosity, and c_p is the material's specified or computed specific heat capacity.

To enable the use of this equation for calculating thermal conductivity, select **kinetic-theory** from the drop-down list to the right of **Thermal Conductivity** in the Create/Edit Materials Dialog Box (p. 2302). The solver will use Equation 7.47 (p. 499) to compute the thermal conductivity.

7.5.4. Composition-Dependent Thermal Conductivity for Multicomponent Mixtures

If you are modeling a flow that includes more than one chemical species (multicomponent flow), you have the option to define a composition-dependent thermal conductivity. (Note that you can also define the thermal conductivity of the mixture as a constant value or a function of temperature, or using kinetic theory.)

To define a composition-dependent thermal conductivity for a mixture, follow these steps:

1. For the mixture material, choose **mass-weighted-mixing-law** or, if you are using the ideal gas law, **ideal-gas-mixing-law** in the drop-down list to the right of **Thermal Conductivity**. If you have a user-defined function that you want to use to model the thermal conductivity, you can choose either the **user-defined** method or the **user-defined-mixing-law** method for the mixture material in the drop-down list.

   The only difference between the **user-defined-mixing-law** and the **user-defined** option for specifying density, viscosity and thermal conductivity of mixture materials, is that with the **user-defined-mixing-**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

law option, the individual properties of the species materials can also be specified. Note that only the constant, the polynomial methods and the user-defined methods are available.

Important

If you use **ideal-gas-mixing-law** for the thermal conductivity of a mixture, you must use **ideal-gas-mixing-law** or **mass-weighted-mixing-law** for viscosity, because these two viscosity specification methods are the only ones that allow specification of the component viscosities, which are used in the ideal gas law for thermal conductivity (Equation 7.48 (p. 500)).

2. Click **Change/Create**.

3. Define the thermal conductivity for each of the fluid materials that comprise the mixture. You may define constant or (if applicable) temperature-dependent thermal conductivities for the individual species. You may also use kinetic theory for the individual thermal conductivities, if applicable.

4. If you selected **user-defined-mixing-law**, define the thermal conductivity for each of the fluid materials that comprise the mixture. You may define constant, or (if applicable) temperature-dependent thermal conductivities, or user-defined thermal conductivities for the individual species. More information about defining properties with user-defined functions can be found in the Fluent Customization Manual.

If you are using the ideal gas law, the solver will compute the mixture thermal conductivity based on kinetic theory as

$$k = \sum_i \frac{X_i k_i}{\sum_j X_j \phi_{ij}} \tag{7.48}$$

where

$$\phi_{ij} = \frac{\left[1 + \left(\frac{\mu_i}{\mu_j}\right)^{1/2} \left(\frac{M_{w,j}}{M_{w,i}}\right)^{1/4}\right]^2}{\left[8\left(1 + \frac{M_{w,i}}{M_{w,j}}\right)\right]^{1/2}} \tag{7.49}$$

and X_i is the mole fraction of species i.

For non-ideal gases, the mixture thermal conductivity is computed based on a simple mass fraction average of the pure species conductivities:

$$k = \sum_i Y_i k_i \tag{7.50}$$

7.5.5. Anisotropic Thermal Conductivity for Solids

The anisotropic conductivity option in ANSYS Fluent solves the conduction equation in solid zones and shells with the thermal conductivity specified as a matrix. The heat flux vector is written as

$$q_i = -k_{ij} \frac{\partial T}{\partial x_j} \tag{7.51}$$

The following options are available for defining anisotropic thermal conductivity in ANSYS Fluent. These are discussed below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- anisotropic
- biaxial (available for shells only)
- orthotropic
- cylindrical orthotropic
- principal axes and principal values
- user-defined anisotropic

Note

When postprocessing solid material properties, the Thermal Conductivity will be displayed as zero if an anisotropic thermal conductivity model is being used for a solid material.

Important

The anisotropic conductivity options are available only with the pressure-based solver; you cannot use them with the density-based solvers.

7.5.5.1. Anisotropic Thermal Conductivity

For anisotropic diffusion, the thermal conductivity matrix (Equation 7.51 (p. 500)) is specified as

$$k_{ij}=k\hat{e}_{ij} \tag{7.52}$$

where k is the conductivity and $\hat{e}_{ij}$ is a matrix (2 × 2 for two dimensions and 3 × 3 for three-dimensional problems). Note that $\hat{e}_{ij}$ can be a non-symmetric matrix.

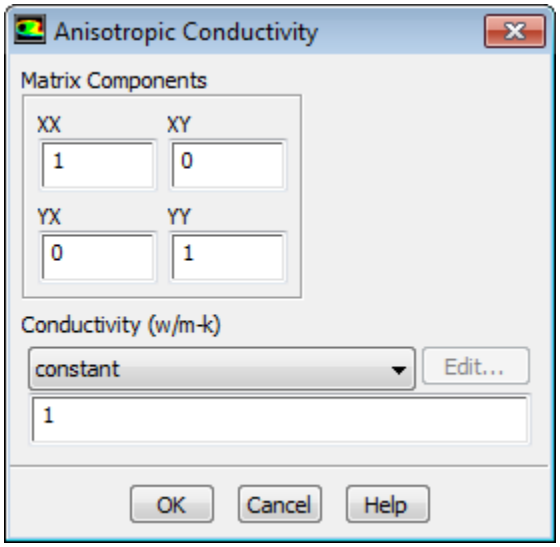
To define anisotropic thermal conductivity for a solid material, select **anisotropic** for **Thermal Conductivity** in the Create/Edit Materials Dialog Box (p. 2302) (Figure 7.20: The Create/Edit Materials Dialog Box (p. 498)). Note the following:

- If the principal axes of your anisotropic material are not aligned with the global coordinate system of your simulation, it may be easier for you to use one of the following options: if the axes are orthogonal, you can use the **orthotropic** option, as described in Orthotropic Thermal Conductivity (p. 503); if the axes are not orthogonal, you can use the **principal-axes-values** option, as described in Principal Axes and Principal Values (p. 506). If you use the **anisotropic** option with unaligned axes, it is your responsibility to transform the thermal conductivity properties into the k_{ij} tensor aligned with the global coordinate system.
- The **anisotropic** option is appropriate when the components of the k_{ij} matrix are constants and do not vary independently; if this is not the case, you can use a UDF to define the matrix and select **user-defined-anisotropic-k**, as described in User-Defined Anisotropic Thermal Conductivity (p. 507).

Selecting **anisotropic** will open the Anisotropic Conductivity Dialog Box (p. 2331) (Figure 7.21: The Anisotropic Conductivity Dialog Box (p. 502)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.21: The Anisotropic Conductivity Dialog Box



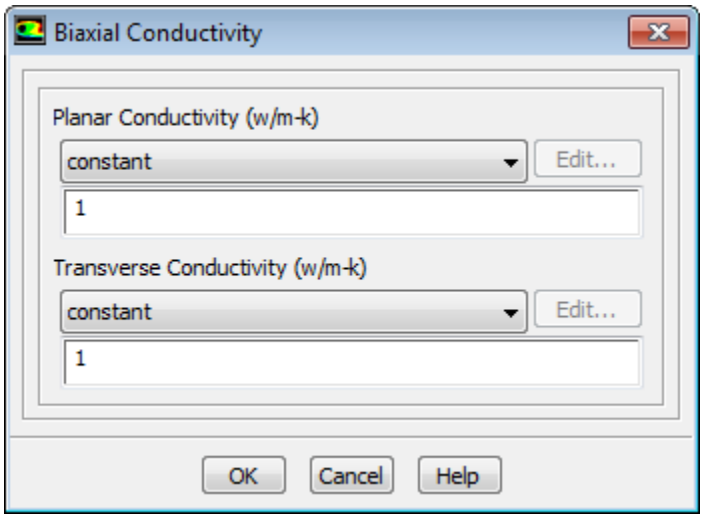
In the Anisotropic Conductivity Dialog Box (p. 2331), enter the **Matrix Components** of matrix $\hat{e}_{ij}$ and then select the **Conductivity** (k in Equation 7.52 (p. 501)) to be a **constant**, polynomial function of temperature (**polynomial**, **piecewise-linear**, **piecewise-polynomial**), or **user-defined** function. See Constant Thermal Conductivity (p. 498) and Thermal Conductivity as a Function of Temperature (p. 498) for details on constants and thermal polynomial functions.

When you select the **user-defined** option, the User-Defined Functions Dialog Box (p. 2321) will open, allowing you to hook a `DEFINE_PROPERTY` UDF *only* if you have previously loaded a compiled UDF library or interpreted the UDF. Otherwise, you will get an error message. Details about user-defined functions can be found in the Fluent Customization Manual.

7.5.5.2. Biaxial Thermal Conductivity

Biaxial thermal conductivity is applicable only for solid materials used for the wall shell conduction model. To define a biaxial thermal conductivity, select **biaxial** in the drop-down list for **Thermal Conductivity** in the Create/Edit Materials Dialog Box (p. 2302). This opens the Biaxial Conductivity Dialog Box (p. 2327) (Figure 7.22: The Biaxial Conductivity Dialog Box (p. 502)).

Figure 7.22: The Biaxial Conductivity Dialog Box



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the Biaxial Conductivity Dialog Box (p. 2327), both the conductivity normal to the surface of the shell (**Transverse Conductivity**) and the conductivity parallel to the surface of the shell (**Planar Conductivity**) can be defined as **constant**, **polynomial**, **piecewise-linear**, or **piecewise-polynomial**. See Constant Thermal Conductivity (p. 498) and Thermal Conductivity as a Function of Temperature (p. 498) for details on these parameters. Note that the **Planar Conductivity** is isotropic. See Figure 12.7: The Shell Conduction Manager Dialog Box (p. 841).

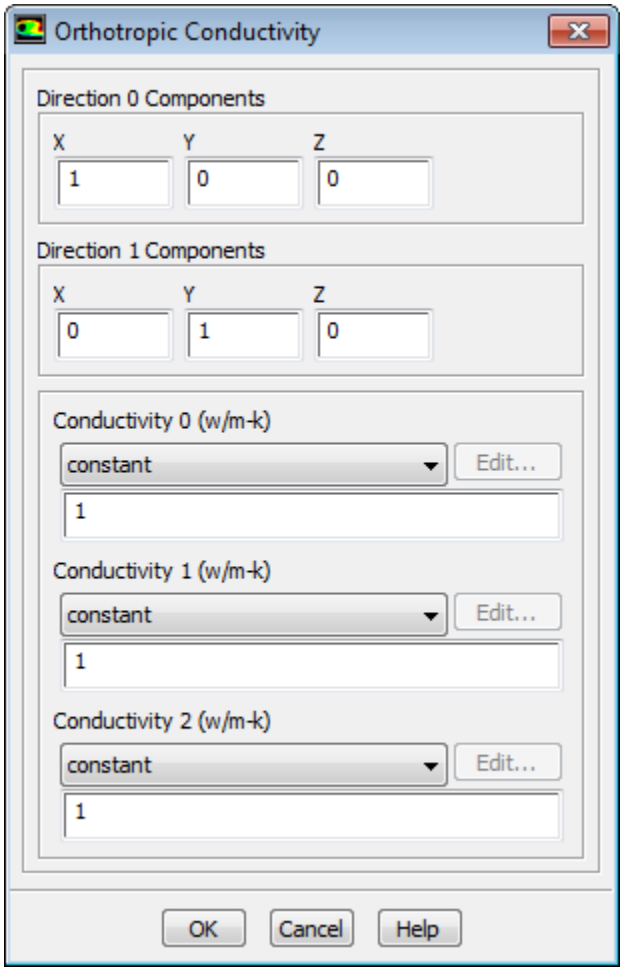
7.5.5.3. Orthotropic Thermal Conductivity

When the orthotropic thermal conductivity is used, the thermal conductivities $\left(k_\xi, k_\eta, k_\zeta\right)$ in the principal directions $\left(\hat{e}_\xi, \hat{e}_\eta, \hat{e}_\zeta\right)$ are specified. The conductivity matrix is then computed as

$$k_{ij} = k_\xi e_{\xi i} e_{\xi j} + k_\eta e_{\eta i} e_{\eta j} + k_\zeta e_{\zeta i} e_{\zeta j} \tag{7.53}$$

To define an orthotropic thermal conductivity in solids, select **orthotropic** in the drop-down list for **Thermal Conductivity** in the Create/Edit Materials Dialog Box (p. 2302). This opens the Orthotropic Conductivity Dialog Box (p. 2330) (Figure 7.23: The Orthotropic Conductivity Dialog Box (p. 503)).

Figure 7.23: The Orthotropic Conductivity Dialog Box



Since the directions $\left(\hat{e}_\xi, \hat{e}_\eta, \hat{e}_\zeta\right)$ are mutually orthogonal, only the first two need to be specified for three-dimensional problems. $\hat{e}_\xi$ is defined using **X, Y, Z** under **Direction 0 Components**, and $\hat{e}_\eta$ is defined using **X, Y, Z** under **Direction 1 Components**. You can define **Conductivity 0** $\left(k_\xi\right)$, **Conduct-**

ivity 1 $\left(k_\eta\right)$, and **Conductivity 2** $\left(k_\zeta\right)$ as **constant**, **polynomial**, **piecewise-linear**, **piecewise-polynomial** functions of temperature, or **user-defined**. See Constant Thermal Conductivity (p. 498) and Thermal Conductivity as a Function of Temperature (p. 498) for details on constant and temperature profile functions.

When you select the **user-defined** option, the User-Defined Functions Dialog Box (p. 2321) will open, allowing you to hook a `DEFINE_PROPERTY` UDF *only* if you have previously loaded a compiled UDF library or interpreted the UDF. Otherwise, you will get an error message. More information about user-defined functions can be found in the Fluent Customization Manual.

Important

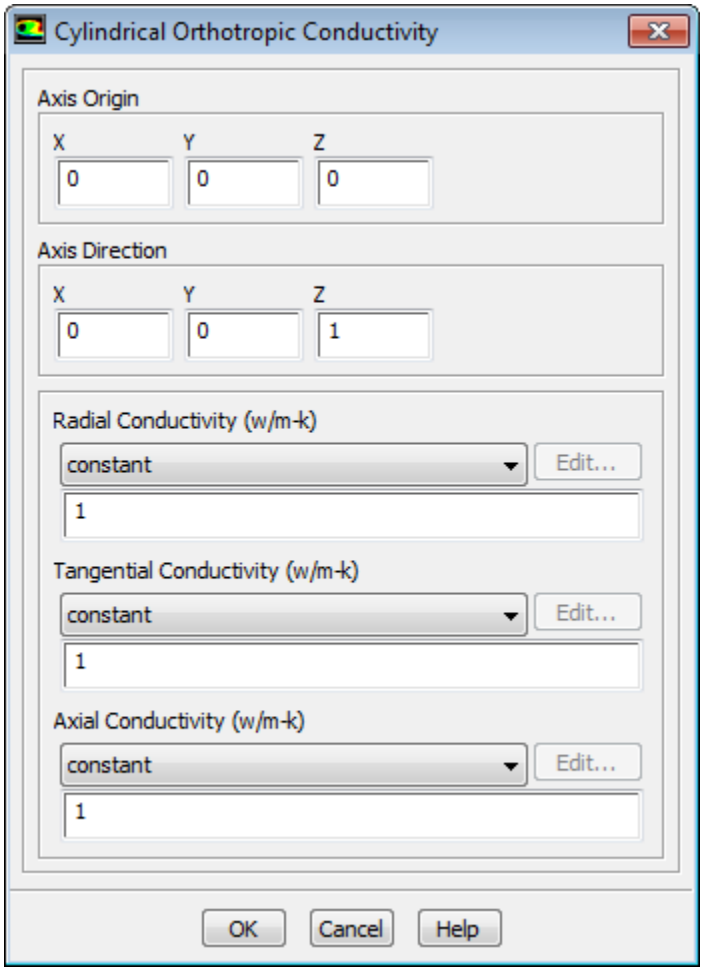
For two-dimensional problems, only the functions $\left(k_\xi, k_\eta\right)$ and the unit vector $\left(\hat{e}_\xi\right)$ need to be specified.

7.5.5.4. Cylindrical Orthotropic Thermal Conductivity

The orthotropic conductivity of solids can be specified in cylindrical coordinates. To define the orthotropic thermal conductivity in cylindrical coordinates, select **cyl-orthotropic** in the drop-down list for **Thermal Conductivity** in the Create/Edit Materials Dialog Box (p. 2302). This opens the Cylindrical Orthotropic Conductivity Dialog Box (p. 2328) (Figure 7.24: The Cylindrical Orthotropic Conductivity Dialog Box (p. 505)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.24: The Cylindrical Orthotropic Conductivity Dialog Box



In three-dimensional cases, the origin and the direction of the cylindrical coordinate system must be specified along with the radial, tangential, and axial direction conductivities. In two-dimensional cases, the origin of the cylindrical coordinate system must be specified along with the radial and tangential direction conductivities. Note that in two-dimensional cases, the direction is always along the +Z axis. ANSYS Fluent will automatically compute the anisotropic conductivity matrix at each cell from this input. The calculation is based on the location of the cell in the cylindrical coordinate system specified.

You can define the **Radial Conductivity**, **Tangential Conductivity**, and **Axial Conductivity** as **constant**, **polynomial**, **piecewise-linear**, **piecewise-polynomial**, or as user-defined functions of temperature. See Constant Thermal Conductivity (p. 498) and Thermal Conductivity as a Function of Temperature (p. 498) for details on constant and thermal profile functions.

When you select the **user-defined** option, the User-Defined Functions Dialog Box (p. 2321) will open, allowing you to hook a `DEFINE_PROPERTY` UDF *only* if you have previously loaded a compiled UDF library or interpreted the UDF. Otherwise, you will get an error message. More information about user-defined functions can be found in the Fluent Customization Manual.

Important

For conductivity calculations near the wall, the cell next to the wall is chosen for computing the conductivity matrix instead of the wall itself.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.5.5.5. Principal Axes and Principal Values

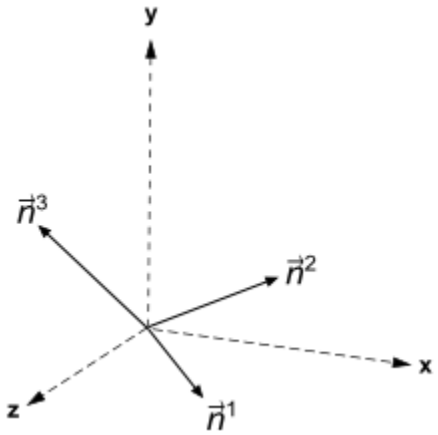
When the principal axes of your anisotropic material are not aligned with the global coordinate system of the simulation, you can define the thermal conductivity using the components of these axes ($\vec{n}^1$, $\vec{n}^2$, and for 3D problems, $\vec{n}^3$) and the associated principle values (K_1, K_2, and for 3D problems, K_3). The axes take the following form:

$$\begin{aligned}\vec{n}^1 &= n_1^1\hat{i} + n_2^1\hat{j} + n_3^1\hat{k} \\ \vec{n}^2 &= n_1^2\hat{i} + n_2^2\hat{j} + n_3^2\hat{k} \\ \vec{n}^3 &= n_1^3\hat{i} + n_2^3\hat{j} + n_3^3\hat{k}\end{aligned} \tag{7.54}$$

Note that while the axes must be linearly independent vectors, they do not have to be unit vectors or orthogonal.

An example of principal axes that are not aligned with the global coordinate system is shown in Figure 7.25: Unaligned Principal Axes (p. 506).

Figure 7.25: Unaligned Principal Axes



Representing the thermal conductivity matrix (k_{ij} in Equation 7.51 (p. 500)) in matrix notation ($[\mathbf{k}]$), the following equation is used by ANSYS Fluent to transform the principal axes and principal values into the correct tensor form:

$$[\mathbf{k}] = [\mathbf{N}][\mathbf{D}][\mathbf{N}]^{-1} \tag{7.55}$$

The matrices in the previous equation are defined as follows:

$$[\mathbf{N}] = \begin{bmatrix} n_1^1 & n_1^2 & n_1^3 \\ n_2^1 & n_2^2 & n_2^3 \\ n_3^1 & n_3^2 & n_3^3 \end{bmatrix} \tag{7.56}$$

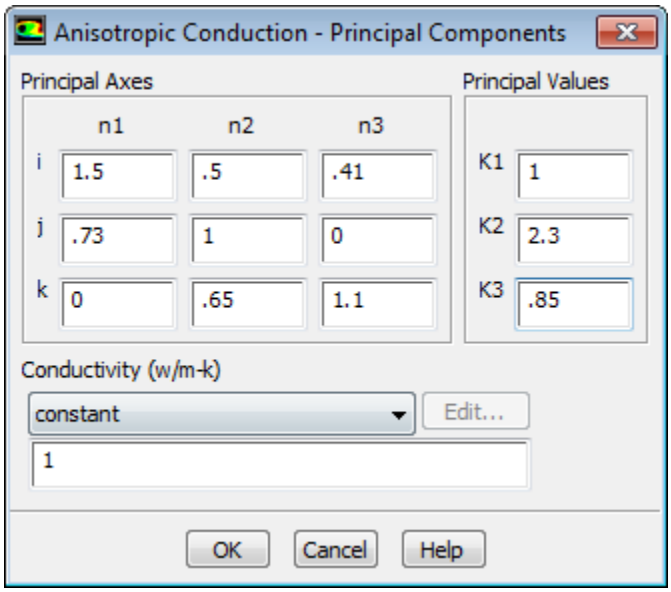
$$[\mathbf{D}] = \begin{bmatrix} kK_1 & 0 & 0 \\ 0 & kK_2 & 0 \\ 0 & 0 & kK_3 \end{bmatrix} \tag{7.57}$$

where k is the conductivity.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To use principal axes and principal values to define the anisotropic thermal conductivity for your solid material, select **principal-axes-values** for **Thermal Conductivity** in the Create/Edit Materials Dialog Box (p. 2302) (Figure 7.20: The Create/Edit Materials Dialog Box (p. 498)). This will open the Anisotropic Conductivity - Principal Components Dialog Box (p. 2331) (Figure 7.26: The Anisotropic Conductivity - Principal Components Dialog Box (p. 507)).

Figure 7.26: The Anisotropic Conductivity - Principal Components Dialog Box



In the **Principal Axes** group box of the Anisotropic Conductivity - Principal Components Dialog Box (p. 2331), enter the **i**, **j**, and **k** components for each of the **n1**, **n2**, and **n3** vectors (Equation 7.54 (p. 506)) that define the principal axes of your material in the global coordinate system; note that these must be linearly independent vectors. Then in the **Principal Values** group box, enter the **K1**, **K2**, and **K3** values that define the magnitudes of the principal values along the associated principal axes. Finally, make a selection from the **Conductivity** drop-down menu to specify k (in Equation 7.57 (p. 506)) as a **constant**, polynomial function of temperature (**polynomial**, **piecewise-linear**, **piecewise-polynomial**), or **user-defined** function, and then define it using the number-entry box that becomes available or the dialog box that opens. See Constant Thermal Conductivity (p. 498) and Thermal Conductivity as a Function of Temperature (p. 498) for details on constants and thermal functions.

When you select the **user-defined** option, the User-Defined Functions Dialog Box (p. 2321) will open, allowing you to hook a `DEFINE_PROPERTY` UDF *only* if you have previously loaded a compiled UDF library or interpreted the UDF. Otherwise, you will get an error message. Details about user-defined functions can be found in the Fluent Customization Manual.

7.5.5.6. User-Defined Anisotropic Thermal Conductivity

You have the option of modeling anisotropic conduction for domains where the components of the thermal conductivity matrix (k_{ij} in Equation 7.52 (p. 501)) are not constants and vary independently, perhaps as a function of space or because the domain has many anisotropic materials stitched together to form a composite. To use this option, you must first define the conductivity matrix using a UDF, as described in `DEFINE_ANISOTROPIC_CONDUCTIVITY` in the Fluent Customization Manual.

After you have interpreted or compiled the `DEFINE_ANISOTROPIC_CONDUCTIVITY` UDF, you must then select **user-defined-anisotropic-k** for **Thermal Conductivity** in the Create/Edit Materials Dialog

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Box (p. 2302). This will open the User-Defined Functions Dialog Box (p. 2321), which you can use to hook the UDF.

7.6. User-Defined Scalar (UDS) Diffusivity

There are two types of UDS diffusivity that you can specify in ANSYS Fluent: isotropic and anisotropic. Diffusion is isotropic when it is the same in all directions. Isotropic diffusion coefficients can be specified in two ways: either as a single **user-defined** that applies to all UDS transport equations defined for your model; or on a per-scalar basis as **constants**, **polynomial** functions of temperature, or **user-defined** functions.

Diffusion is anisotropic when the diffusion coefficients are different in different directions. Anisotropic diffusion can be specified by a tensor diffusion coefficient matrix Γ (Equation 7.58 (p. 508)) for each UDS (in both fluid and solid zones) in four different ways: general **anisotropic**, **orthotropic**, **cyl-orthotropic**, and **user-defined-anisotropic**. All UDS diffusivity parameters are set from the Create/Edit Materials Dialog Box (p. 2302) and are discussed below. Note that details about how to define and use UDFs in UDS transport equations is discussed in the Fluent Customization Manual.

The second-order diffusion term in the most general form is

$$\nabla \cdot \left(\Gamma \cdot \nabla \phi^k \right) \tag{7.58}$$

where Γ is a 3 × 3 tensor in 3D.

For additional information, see the following sections:

7.6.1. Isotropic Diffusion
7.6.2. Anisotropic Diffusion
7.6.3. User-Defined Anisotropic Diffusivity

7.6.1. Isotropic Diffusion

For isotropic diffusion, Γ in Equation 7.58 (p. 508) is equal to a scalar Γ times the identity matrix and the equation reduces to

$$\nabla \cdot \left(\Gamma \nabla \phi^k \right) \tag{7.59}$$

You can specify isotropic diffusivity as a single user-defined function that applies to all UDS transport equations. For this case, choose **user-defined** from the drop-down list for **UDS Diffusivity** in the Create/Edit Materials Dialog Box (p. 2302).

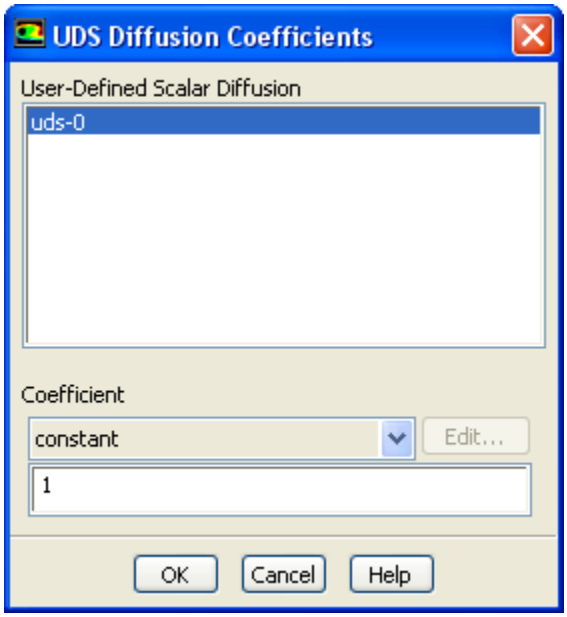
Setup → **Materials**

If you have previously loaded a compiled UDF library or have interpreted the UDF, then the User-Defined Functions Dialog Box (p. 2321) will open, allowing you to hook the `DEFINE_DIFFUSIVITY` UDF to ANSYS Fluent. If no functions have been loaded, you will get an error message. More information about user-defined functions can be found in the Fluent Customization Manual.

Isotropic diffusion coefficients can also be defined on a per-scalar basis by selecting **defined-per-uds** from the drop-down list for **UDS Diffusivity** in the Create/Edit Materials Dialog Box (p. 2302). This will open the UDS Diffusion Coefficients Dialog Box (p. 2347) (Figure 7.27: The UDS Diffusion Coefficients Dialog Box (p. 509)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.27: The UDS Diffusion Coefficients Dialog Box



In the UDS Diffusion Coefficients Dialog Box (p. 2347), select a scalar equation (for example, **uds-0**) and then choose a **constant**, **polynomial**, or **user-defined** function from the **Coefficient** drop-down list. For the default fluid (air), the constant diffusion coefficient is 1 kg/m-s. If you choose **polynomial**, the Polynomial Profile Dialog Box (p. 2318) will open and you can specify your coefficients as a function of temperature. See Inputs for Polynomial Functions (p. 475) for details.

When you select the **user-defined** option, the User-Defined Functions Dialog Box (p. 2321) will open, allowing you to hook a `DEFINE_DIFFUSIVITY` UDF *only* if you have previously loaded a compiled UDF library or interpreted a UDF. Otherwise, you will get an error message. More information about user-defined functions can be found in the Fluent Customization Manual.

7.6.2. Anisotropic Diffusion

You can specify anisotropic diffusion coefficients in both fluid and solid zones by defining the tensor diffusion coefficient matrix Γ (Equation 7.58 (p. 508)) on a per-scalar basis. You can use anisotropic diffusivity for UDS scalar transport equations to model species transport equations in porous media and in solids where species diffusion shows anisotropic behavior.

Important

- Note that the anisotropic diffusion options discussed in the following sections are available with the pressure-based solver and the density-based solvers.
- UDS diffusion coefficients can be postprocessed only in those cells that have isotropic diffusivity. In all other cells, the diffusion coefficient will be zero.

In all cases, you enable anisotropic diffusion by selecting **defined-per-uds** under **UDS Diffusivity** in the Create/Edit Materials Dialog Box (p. 2302). This will open the UDS Diffusion Coefficients Dialog Box (p. 2347) (Figure 7.27: The UDS Diffusion Coefficients Dialog Box (p. 509)).

Setup → Materials

In the UDS Diffusion Coefficients Dialog Box (p. 2347), select a scalar equation (for example, **uds-0**) and then choose one of the following methods under **Coefficient** to specify the anisotropic diffusion coefficient. These methods are described in detail below.

- anisotropic
- orthotropic
- cylindrical orthotropic
- user-defined anisotropic

7.6.2.1. Anisotropic Diffusivity

For anisotropic diffusivity, you can specify Γ in Equation 7.58 (p. 508) in the form $K\mathbf{\Gamma}$ where K is a constant $3x3$ matrix in 3D and Γ is a scalar multiplier.

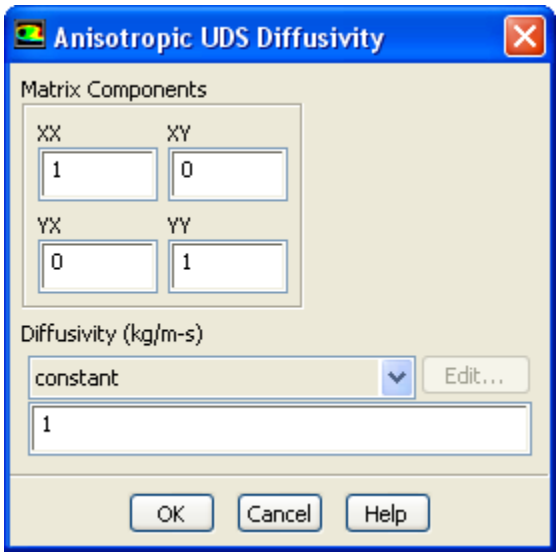
The diffusion coefficient matrix is specified as

$$k_{ij}=k\hat{e}_{ij} \tag{7.60}$$

where k is the diffusivity and $\hat{e}_{ij}$ is a matrix (2 × 2 for two dimensions and 3 × 3 for three-dimensional problems). Note that $\hat{e}_{ij}$ can be a non-symmetric matrix.

To specify anisotropic diffusion coefficients, first select a scalar equation (for example, **uds-0**) from the **User-Defined Scalar Diffusion** list in the UDS Diffusion Coefficients Dialog Box (p. 2347) (Figure 7.27: The UDS Diffusion Coefficients Dialog Box (p. 509)). Then choose **anisotropic** in the drop-down list under **Coefficient**. This will open the **Anisotropic UDS Diffusivity** dialog box (Figure 7.28: The Anisotropic UDS Diffusivity Dialog Box (p. 510)).

Figure 7.28: The Anisotropic UDS Diffusivity Dialog Box



In the **Anisotropic UDS Diffusivity** dialog box, enter the **Matrix Components** and then select the **Diffusivity** to be a **constant**, polynomial function of temperature (**polynomial**, **piecewise-linear**, **piecewise-polynomial**), or **user-defined**. See Inputs for Polynomial Functions (p. 475), Inputs for Piecewise-Linear Functions (p. 476), and Inputs for Piecewise-Polynomial Functions (p. 478) for details on polynomial temperature functions.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you select the **user-defined** option, the User-Defined Functions Dialog Box (p. 2321) will open, allowing you to hook a `DEFINE_DIFFUSIVITY` UDF *only* if you have previously loaded a compiled UDF library or interpreted a UDF. Otherwise, you will get an error message. More information about user-defined functions can be found in the Fluent Customization Manual.

7.6.2.2. Orthotropic Diffusivity

For orthotropic diffusivity, you can specify Γ in Equation 7.58 (p. 508) through 'principal' direction vectors and diffusion coefficients along these directions. ANSYS Fluent, in turn, computes Γ from parameters that you supply. The principal directions are the same everywhere, but each of he directional diffusion coefficients can be specified as a constant, polynomial function of temperature, or through user-defined functions.

When orthotropic diffusivity is used, the diffusion coefficients $\left(k_\xi, k_\eta, k_\zeta\right)$ in the principal directions $\left(\hat{e}_\xi, \hat{e}_\eta, \hat{e}_\zeta\right)$ are specified. The diffusivity matrix is then computed as

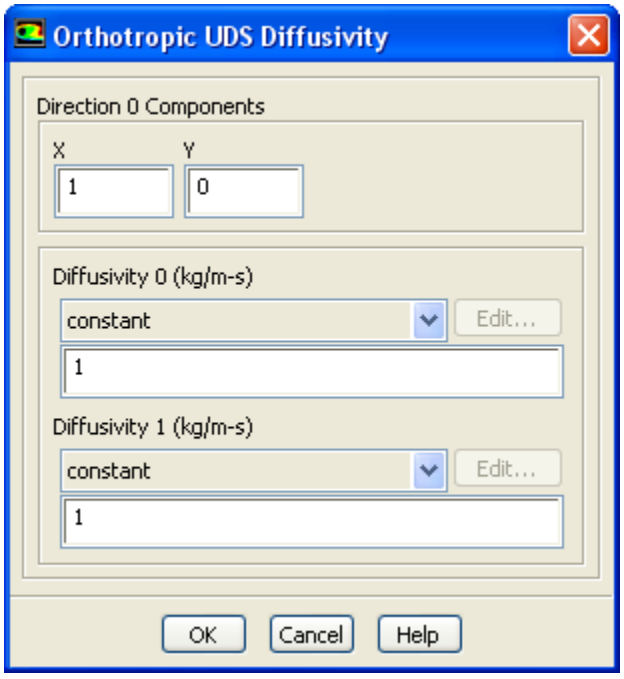
$$k_{ij} = k_\xi e_{\xi i} e_{\xi j} + k_\eta e_{\eta i} e_{\eta j} + k_\zeta e_{\zeta i} e_{\zeta j} \tag{7.61}$$

Important

For two-dimensional problems, only the functions $\left(k_\xi, k_\eta\right)$ and the unit vector $\left(\hat{e}_\xi\right)$ need to be specified.

To specify orthotropic diffusion coefficients, first select a scalar equation (for example, **uds-0**) from the **User-Defined Scalar Diffusion** list in the UDS Diffusion Coefficients Dialog Box (p. 2347) (Figure 7.27: The UDS Diffusion Coefficients Dialog Box (p. 509)). Then choose **orthotropic** in the drop-down list under **Coefficient**. This will open the **Orthotropic UDS Diffusivity** dialog box (Figure 7.29: The Orthotropic UDS Diffusivity Dialog Box (p. 511)).

Figure 7.29: The Orthotropic UDS Diffusivity Dialog Box



Since the directions $\left(\hat{e}_\xi, \hat{e}_\eta, \hat{e}_\zeta\right)$ are mutually orthogonal, only the first two need to be specified for three-dimensional problems. $\hat{e}_\xi$ is defined using **X,Y,Z** under **Direction 0 Components**, and $\hat{e}_\eta$ is defined using **X,Y,Z** under **Direction 1 Components**. You can define **Diffusivity 0** $\left(k_\xi\right)$, **Diffusivity 1** $\left(k_\eta\right)$, and **Diffusivity 2** $\left(k_\zeta\right)$ as **constant**, **polynomial**, **piecewise-linear**, **piecewise-polynomial** functions of temperature, or **user-defined**. See Inputs for Polynomial Functions (p. 475), Inputs for Piecewise-Linear Functions (p. 476), and Inputs for Piecewise-Polynomial Functions (p. 478) for details on polynomial temperature functions.

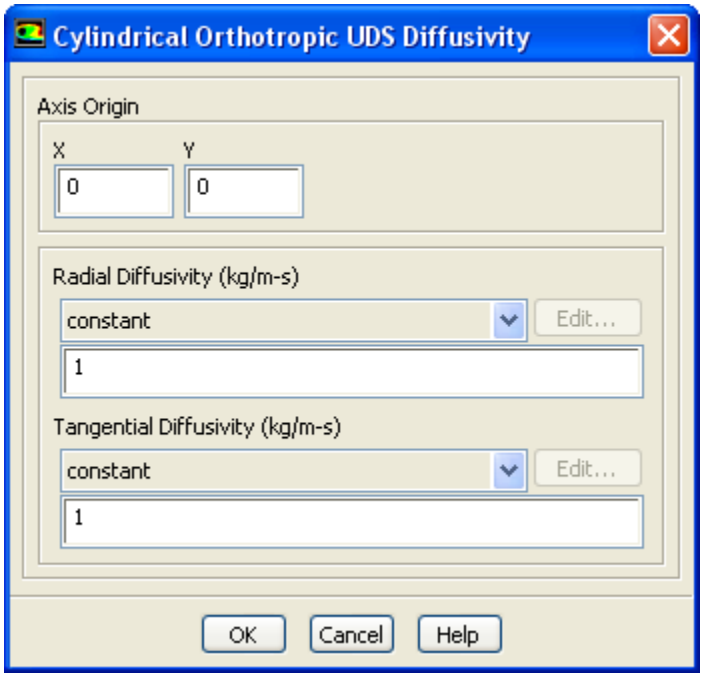
When you select the **user-defined** option, the User-Defined Functions Dialog Box (p. 2321) will open, allowing you to hook a `DEFINE_DIFFUSIVITY` UDF *only* if you have previously loaded a compiled UDF library or interpreted a UDF. If no functions have been loaded, you will get an error message. More information about user-defined functions can be found in the Fluent Customization Manual.

7.6.2.3. Cylindrical Orthotropic Diffusivity

Orthotropic UDS diffusivity can also be specified on a per-scalar basis in cylindrical coordinates. This method is similar to orthotropic UDS diffusivity, except that the principal directions are specified as radial, tangential, and axial.

To specify cylindrical orthotropic diffusion coefficients, first select a scalar equation (for example, **uds-0**) from the **User-Defined Scalar Diffusion** list in the UDS Diffusion Coefficients Dialog Box (p. 2347) (Figure 7.27: The UDS Diffusion Coefficients Dialog Box (p. 509)). Then choose **cyl-orthotropic** in the drop-down list under **Coefficient**. This will open the **Cylindrical Orthotropic UDS Diffusivity** dialog box (Figure 7.30: The Cylindrical Orthotropic UDS Diffusivity Dialog Box (p. 512)).

Figure 7.30: The Cylindrical Orthotropic UDS Diffusivity Dialog Box



In three-dimensional cases, the origin and the direction of the cylindrical coordinate system must be specified along with the radial, tangential, and axial direction conductivities. In two-dimensional cases, the origin of the cylindrical coordinate system must be specified along with the radial and tangential direction conductivities. Note that in two-dimensional cases, the direction is always along the +z axis.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS Fluent will automatically compute the anisotropic diffusivity matrix at each cell from this input. The calculation is based on the location of the cell in the cylindrical coordinate system specified.

You can define the **Radial Diffusivity**, **Tangential Diffusivity**, and **Axial Diffusivity** as **constant**, **polynomial**, **piecewise-linear**, **piecewise-polynomial**, or as **user-defined** functions of temperature, using the drop-down list below each of the diffusivities. See Inputs for Polynomial Functions (p. 475), Inputs for Piecewise-Linear Functions (p. 476), and Inputs for Piecewise-Polynomial Functions (p. 478) for details on polynomial temperature functions.

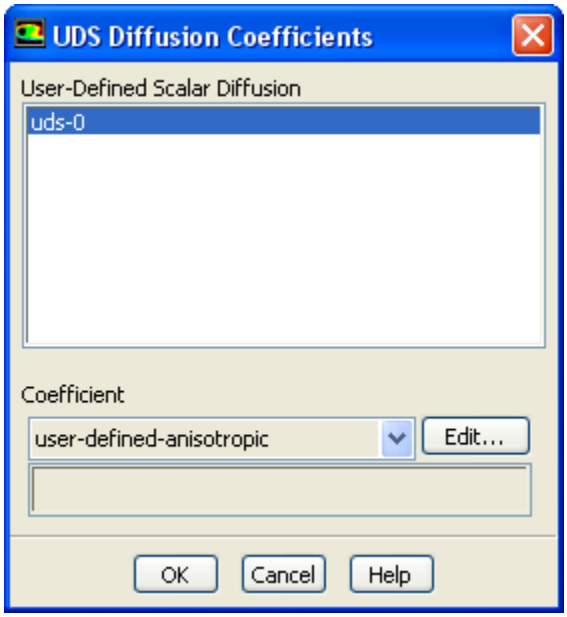
When you select the **user-defined** option, the User-Defined Functions Dialog Box (p. 2321) will open, allowing you to hook a `DEFINE_DIFFUSIVITY` UDF *only* if you have previously loaded a compiled UDF library or interpreted a UDF. If no functions have been loaded, you will get an error message. More information about user-defined functions can be found in the Fluent Customization Manual.

7.6.3. User-Defined Anisotropic Diffusivity

You can specify Γ in Equation 7.58 (p. 508) on a per-scalar basis, directly, through user-defined functions (UDFs).

To specify a UDF for anisotropic diffusivity on a per-scalar basis, first select a scalar equation (for example, **uds-0**) from the **User-Defined Scalar Diffusion** list in the UDS Diffusion Coefficients Dialog Box (p. 2347) (Figure 7.31: The UDS Diffusion Coefficients Dialog Box (p. 513)).

Figure 7.31: The UDS Diffusion Coefficients Dialog Box



Then choose **user-defined-anisotropic** in the drop-down list under **Coefficient**. The User-Defined Functions Dialog Box (p. 2321) will open, allowing you to hook a `DEFINE_ANISOTROPIC_DIFFUSIVITY` UDF *only* if you have previously loaded a compiled UDF library or interpreted a UDF. Otherwise, you will get an error message. More information about user-defined functions can be found in the Fluent Customization Manual.

7.7. Specific Heat Capacity

The specific heat capacity must be defined when the energy equation is active. ANSYS Fluent provides several options for definition of the heat capacity:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- constant heat capacity
- temperature- and/or composition-dependent heat capacity
- kinetic theory

Each of these input options and the governing physical models are detailed in this section. In all cases, you will define the **Cp** in the Create/Edit Materials Dialog Box (p. 2302).

Setup → **Materials**

Specific heat capacity is input in units of J/kg-K in SI units or BTU/lbm-°R in British units.

Important

For combustion applications, a temperature-dependent specific heat is recommended.

For additional information, see the following sections:
7.7.1. Input of Constant Specific Heat Capacity
7.7.2. Specific Heat Capacity as a Function of Temperature
7.7.3. Defining Specific Heat Capacity Using Kinetic Theory
7.7.4. Specific Heat Capacity as a Function of Composition

7.7.1. Input of Constant Specific Heat Capacity

If you want to define the heat capacity as a constant, check that **constant** is selected in the drop-down list to the right of **Cp** in the Create/Edit Materials Dialog Box (p. 2302), and enter the value of heat capacity.

The specific heat for the default fluid (air) is 1006.43 J/kg-K.

7.7.2. Specific Heat Capacity as a Function of Temperature

You can also choose to define the specific heat capacity as a function of temperature. Three types of functions are available:

- piecewise-linear:

$$c_p(T) = c_{p_n} + \frac{c_{p_{n+1}} - c_{p_n}}{T_{n+1} - T_n}(T - T_n) \tag{7.62}$$

- piecewise-polynomial:

$$\begin{aligned} &\text{for } T_{min,1} \le T < T_{max,1}: c_p(T) = A_1 + A_2 T + A_3 T^2 + \ldots \\ &\text{for } T_{min,2} \le T < T_{max,2}: c_p(T) = B_1 + B_2 T + B_3 T^2 + \ldots \end{aligned} \tag{7.63}$$

- polynomial:

$$c_p(T) = A_1 + A_2 T + A_3 T^2 + \ldots \tag{7.64}$$

You can input the data pairs (T_n, c_{p_n}), ranges and coefficients A_i and B_i, or coefficients A_i that describe these functions using the Create/Edit Materials Dialog Box (p. 2302), as described in Defining Properties Using Temperature-Dependent Functions (p. 475).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.7.3. Defining Specific Heat Capacity Using Kinetic Theory

If you are using the ideal gas law (as described in Density (p. 479)), you have the option to define the specific heat capacity using kinetic theory as

$$c_{p,i}=\frac{1}{2}\frac{R}{M_{w,i}}\left(f_i+2\right) \tag{7.65}$$

where f_i is the number of modes of energy storage (degrees of freedom) for the gas species i that you can input by selecting **kinetic-theory** from the drop-down list to the right of **Cp** in the Create/Edit Materials Dialog Box (p. 2302). The solver will use your kinetic theory inputs in Equation 7.65 (p. 515) to compute the specific heat capacity. See Kinetic Theory Parameters (p. 529) for details about kinetic theory inputs.

7.7.4. Specific Heat Capacity as a Function of Composition

If you are modeling a flow that includes more than one chemical species (multicomponent flow), you have the option to define a composition-dependent specific heat capacity. You can also define the heat capacity of the mixture as a constant value or a function of temperature, or using kinetic theory.

To define a composition-dependent specific heat capacity for a mixture, follow these steps:

1. For the mixture material, choose **mixing-law** in the drop-down list to the right of **Cp**.
2. Click **Change/Create**.
3. Define the specific heat capacity for each of the fluid materials that comprise the mixture. You may define constant or (if applicable) temperature-dependent heat capacities for the individual species. You may also use kinetic theory for the individual heat capacities, if applicable.

The solver will compute the mixture's specific heat capacity as a mass fraction average of the pure species heat capacities:

$$c_p=\sum_i Y_i c_{p,i} \tag{7.66}$$

7.8. Radiation Properties

When you have activated one of the radiation models (except for the surface-to-surface model, which requires no additional properties), there will be additional properties for you to set in the Create/Edit Materials Dialog Box (p. 2302):

- For the P-1 model, you must set the radiation **Absorption Coefficient** and **Scattering Coefficient** (a and σ_s in Equation 5.18 in the Theory Guide).
- For the Rosseland radiation model, you will also need to set the **Absorption Coefficient** and **Scattering Coefficient** (a and σ_s in Equation 5.19 in the Theory Guide).
- For the DTRM, only the **Absorption Coefficient** is required (a in Equation 5.52 in the Theory Guide).
- For the DO model set both, the **Absorption Coefficient** and the **Scattering Coefficient** (a and σ_s in Equation 5.59 in the Theory Guide). In addition, if you are modeling semi-transparent media, specify the **Refractive Index** (n_a or n_b in Equation 5.78 in the Theory Guide). With the DO model, you can specify radiation properties for solid materials, to be used when semi-transparent media are modeled.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For additional information, see the following sections:

7.8.1. Absorption Coefficient
7.8.2. Scattering Coefficient
7.8.3. Refractive Index
7.8.4. Reporting the Radiation Properties

7.8.1. Absorption Coefficient

To define the absorption coefficient, you can specify a constant value, a temperature-dependent function (see Defining Properties Using Temperature-Dependent Functions (p. 475)), a composition-dependent function, or a user-defined function. The absorbing and emitting parts of the radiative transfer equation (RTE), Equation 5.17 in the Theory Guide, is a function of the absorption coefficient. The absorbing or emitting effects depend on the chosen radiation model. If there are only absorption effects, then Lambert's Law of absorption applies

$$I = I_o \exp\left(-ax\right) \tag{7.67}$$

where I is the radiation intensity, a is the absorption coefficient, and x is the distance through the material.

If you are modeling non-gray radiation with the P-1 or the DO radiation model, you also have the option to specify a constant absorption coefficient in each of the gray bands. The absorption coefficient is requested in units of 1/length. Along with the scattering coefficient, it describes the change in radiation intensity per unit length along the path through the fluid medium. Absorption coefficients can be computed using tables of emissivity for CO_2 and H_2O, which are generally available in textbooks on radiation heat transfer.

7.8.1.1. Inputs for a Constant Absorption Coefficient

To define a constant absorption coefficient, simply enter the value in the field next to **Absorption Coefficient** in the Create/Edit Materials Dialog Box (p. 2302). Select **constant** in the drop-down list first if it is not already selected.

7.8.1.2. Inputs for a Composition-Dependent Absorption Coefficient

ANSYS Fluent also allows you to input a composition-dependent absorption coefficient, where the local value of a is a function of the local mass fractions of water vapor and carbon dioxide. This modeling option can be useful for the simulation of radiation in combustion applications. The variable-absorption-coefficient model used by ANSYS Fluent is the weighted-sum-of-gray-gases model (WSGGM) described in Radiation in Combusting Flows in the Theory Guide. To activate it, first enable the species calculation and make sure that CO_2 and H_2O are present in the mixture. Next, select **wsggm-domain-based**, **wsggm-user-specified**, or **user-defined-wsggm** in the drop-down list to the right of **Absorption Coefficient** in the **Create/Edit Materials** dialog box. If you select **user-defined-wsggm**, the **User-Defined Functions** dialog box will open, allowing you to select a previously loaded compiled UDF library or a previously interpreted UDF (see Hooking `DEFINE_WSGGM_ABS_COEFF` UDFs in the Fluent Customization Manual). The WSGGM options differ in the method used to compute the path length, as described in the section that follows.

7.8.1.2.1. Path Length Inputs

When the WSGGM is used to compute the absorption coefficient, you can choose how path length, s, is defined for Equation 5.106 in the Theory Guide. See Radiation in Combusting Flows in the Theory Guide to determine which method is appropriate for your case.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You will select the path length method when you choose the property input method for **Absorption Coefficient**, as described previously.

- If you choose **wsggm-domain-based**, s is set equal to a mean beam length calculated by ANSYS Fluent according to Equation 5.107 in the Theory Guide, which is an average dimension of the domain; no further inputs are required.
- If you choose **wsggm-user-specified**, s is set equal to a mean beam length that you enter for **Path Length** in the WSGGM User Specified Dialog Box (p. 2348).
- If you choose **user-defined-wsggm**, ANSYS Fluent will initially compute the absorption coefficient in the same manner as described for the **wsggm-domain-based** option; however, you have the option of writing a user-defined function that customizes this calculated value. If the soot model is enabled, you can also use the UDF to customize the soot absorption coefficient computed by ANSYS Fluent. See `DEFINE_WSGGM_ABS_COEFF` in the Fluent Customization Manual for further details.

7.8.1.2.1.1. Inputs for a Non-Gray Radiation Absorption Coefficient

If you are using the non-gray DO model (see The DO Model Equations of the Theory Guide and Defining Non-Gray Radiation for the DO Model (p. 865)) or the non-gray P-1 model (see The P-1 Model Equations of the Theory Guide and Setting Up the P-1 Model with Non-Gray Radiation (p. 847)), you can specify a different constant absorption coefficient for each of the bands used by the gray-band model. Select **gray-band** from the **Absorption Coefficient** drop-down list in the **Create/Edit Materials** dialog box and then define the absorption coefficient for each band in the Gray-Band Absorption Coefficient Dialog Box (p. 2349). (Note that you must complete this dialog box in order to proceed.)

7.8.1.2.1.2. Effect of Particles and Soot on the Absorption Coefficient

ANSYS Fluent will include the effect of particles on the absorption coefficient if you have turned on the **Particle Radiation Interaction** option in the Discrete Phase Model Dialog Box (p. 2278) (only for the P-1 and DO radiation models).

If you are modeling soot formation and you want to include the effect of soot formation on the absorption coefficient, turn on the **Soot-Radiation Interaction** in the Soot Model Dialog Box (p. 2262). The soot effects can be included for any of the radiation models, as long as you are using the WSGGM to compute a composition-dependent absorption coefficient. Note that you can use the **user-defined-wsggm** option to customize the soot absorption coefficient calculated by ANSYS Fluent, as described previously.

7.8.2. Scattering Coefficient

The scattering coefficient is, by default, set to zero, and it is assumed to be isotropic. You can specify a constant value, a temperature-dependent function (see Defining Properties Using Temperature-Dependent Functions (p. 475)), or a user-defined function. You can also specify a non-isotropic phase function.

The scattering coefficient is requested in units of 1/length. Along with the absorption coefficient, it describes the change in radiation intensity per unit length along the path through the fluid medium. You may want to increase the scattering coefficient in combustion systems, where particulates may be present.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.8.2.1. Inputs for a Constant Scattering Coefficient

To define a constant scattering coefficient, simply enter the value in the field next to **Scattering Coefficient** in the Create/Edit Materials Dialog Box (p. 2302). (Select **constant** in the drop-down list first if it is not already selected.)

7.8.2.2. Inputs for the Scattering Phase Function

Scattering is assumed to be isotropic, by default, but you can also specify a linear-anisotropic scattering function. If you are using the DO model, Delta-Eddington and user-defined scattering functions are also available.

7.8.2.2.1. Isotropic Phase Function

To model isotropic scattering, select **isotropic** in the **Scattering Phase Function** drop-down list. No further inputs are necessary. This is the default setting in ANSYS Fluent.

7.8.2.2.2. Linear-Anisotropic Phase Function

To model anisotropic scattering, select **linear-anisotropic** in the **Scattering Phase Function** drop-down list and set the value of the phase function coefficient (C in Equation 5.19 in the Theory Guide).

7.8.2.2.3. Delta-Eddington Phase Function

To use a Delta-Eddington phase function, select **delta-eddington** in the **Scattering Phase Function** drop-down list. This will open the Delta-Eddington Scattering Function Dialog Box (p. 2349), in which you can specify the **Forward Scattering Factor** and **Asymmetry Factor** (f and C in Equation 5.68 in the Theory Guide). Since this is a modal dialog box, you must tend to it immediately.

7.8.2.2.4. User-Defined Phase Function

To use a user-defined phase function, select **user-defined** in the **Scattering Phase Function** drop-down list. The user-defined function will contain specifications for Φ^* and f in Equation 5.69 in the Theory Guide. More information about user-defined functions can be found in the Fluent Customization Manual.

7.8.3. Refractive Index

The refractive index is the ratio of speed of light in the medium to the speed of light in vacuum. It is by default set to 1. You can specify a constant value in the field next to **Refractive Index**.

If you are using the non-gray DO model (see The DO Model Equations of the Theory Guide and Defining Non-Gray Radiation for the DO Model (p. 865)) or the non-gray P-1 model (see The P-1 Model Equations of the Theory Guide and Setting Up the P-1 Model with Non-Gray Radiation (p. 847)), you can specify a different constant refractive index for each of the bands used by the gray-band model. Select **refractive-band** from the **Refractive Index** drop-down list in the **Create/Edit Materials** dialog box and then define the refractive index for each band in the Gray-Band Refractive Index Dialog Box (p. 2350). Note that because this is a modal dialog box, you must tend to it immediately.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.8.4. Reporting the Radiation Properties

You can display the computed local values for a and σ_s using the **Absorption Coefficient** and **Scattering Coefficient** items in the **Radiation...** category of the variable selection drop-down list that appears in postprocessing dialog boxes. You will also find the **Refractive Index** in the **Radiation...** category.

7.9. Mass Diffusion Coefficients

For species transport calculations, there are two ways to model the diffusion of chemical species. For most applications the Fick's law approximation is adequate, but for some applications (for example, diffusion-dominated laminar flows such as chemical vapor deposition), the full multicomponent diffusion model is recommended.

Important

The full multicomponent diffusion model is enabled in the Species Model Dialog Box (p. 2216) and is computationally expensive.

For additional information, see the following sections:

7.9.1. Fickian Diffusion
7.9.2. Full Multicomponent Diffusion
7.9.3. Anisotropic Species Diffusion
7.9.4. Thermal Diffusion Coefficients
7.9.5. Mass Diffusion Coefficient Inputs
7.9.6. Mass Diffusion Coefficient Inputs for Turbulent Flow

7.9.1. Fickian Diffusion

Mass diffusion coefficients are required whenever you are solving species transport equations in multicomponent flows. Mass diffusion coefficients are used to compute the diffusion flux of a chemical species in a laminar flow using (by default) Fick's law:

$$J_i = -\rho D_{i,m} \nabla Y_i - D_{T,i} \frac{\nabla T}{T} \tag{7.68}$$

where $D_{i,m}$ is the mass diffusion coefficient for species i in the mixture and $D_{T,i}$ is the thermal (Soret) diffusion coefficient.

Equation 7.68 (p. 519) is strictly valid when the mixture composition is not changing, or when $D_{i,m}$ is independent of composition. This is an acceptable approximation in dilute mixtures when $Y_i << 1$, for all i except the carrier gas. ANSYS Fluent can also compute the transport of non-dilute mixtures in laminar flows by treating such mixtures as multicomponent systems. Within ANSYS Fluent, $D_{i,m}$ can be specified in a variety of ways, including by specifying $\mathcal{D}_{ij}$, the binary mass diffusion coefficient of component i in component j. $\mathcal{D}_{ij}$ is not used directly, however; instead, the diffusion coefficient in the mixture, $D_{i,m}$, is computed as

$$D_{i,m} = \frac{1 - X_i}{\sum_{j, j \neq i} \left(X_j / \mathcal{D}_{ij} \right)} \tag{7.69}$$

where X_i is the mole fraction of species i. You can input $D_{i,m}$ or $\mathcal{D}_{ij}$ for each chemical species, as described in Mass Diffusion Coefficient Inputs (p. 524).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In turbulent flows, Equation 7.68 (p. 519) is replaced with the following form:

$$J_i = -\left(\rho D_{i,m} + \frac{\mu_t}{Sc_t}\right)\nabla Y_i - D_{T,i}\frac{\nabla T}{T} \tag{7.70}$$

where Sc_t is the effective Schmidt number for the turbulent flow:

$$Sc_t = \frac{\mu_t}{\rho D_t} \tag{7.71}$$

and D_t is the effective mass diffusion coefficient due to turbulence.

In turbulent flows your mass diffusion coefficient inputs consist of defining the molecular contribution to diffusion $D_{i,m}$ using the same methods available for the laminar case, with the added option to alter the default settings for the turbulent Schmidt number. As seen from Equation 7.71 (p. 520), this parameter relates the effective mass diffusion coefficient due to turbulence with the eddy viscosity μ_t. As discussed in Mass Diffusion Coefficient Inputs for Turbulent Flow (p. 527), the turbulent diffusion coefficient normally overwhelms the laminar diffusion coefficient, so the default constant value for the laminar diffusion coefficient is usually acceptable.

7.9.2. Full Multicomponent Diffusion

A careful treatment of chemical species diffusion in the species transport and energy equations is important when details of the molecular transport processes are significant (for example, in diffusion-dominated laminar flows). As one of the laminar-flow diffusion models, ANSYS Fluent has the ability to model full multicomponent species transport.

7.9.2.1. General Theory

For multicomponent systems it is not possible, in general, to derive relations for the diffusion fluxes containing the gradient of only one component (as described in Fickian Diffusion (p. 519)). Here, the Maxwell-Stefan equations will be used to obtain the diffusive mass flux. This will lead to the definition of generalized Fick's law diffusion coefficients [120] (p. 2893). This method is preferred over computing the multicomponent diffusion coefficients since their evaluation requires the computation of N^2 cofactor determinants of size $(N-1)\times(N-1)$, and one determinant of size $N\times N$ [115] (p. 2893), where N is the number of chemical species.

7.9.2.2. Maxwell-Stefan Equations

From Merk [71] (p. 2890), the Maxwell-Stefan equations can be written as

$$\sum_{\substack{j=1\\ j\neq i}}^{N}\frac{X_i X_j}{\mathcal{D}_{ij}}\left(\vec{V}_j - \vec{V}_i\right) = \vec{d}_i - \frac{\nabla T}{T}\sum_{\substack{j=1\\ j\neq i}}^{N}\frac{X_i X_j}{\mathcal{D}_{ij}}\left(\frac{D_{T,j}}{\rho_j} - \frac{D_{T,i}}{\rho_i}\right) \tag{7.72}$$

where,

X = the mole fraction

$\vec{V}$ = the diffusion velocity

$\mathcal{D}_{ij}$ = the binary mass diffusion coefficient of species i in species j

D_T = the thermal diffusion coefficient.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For an ideal gas the Maxwell diffusion coefficients are equal to the binary diffusion coefficients. If the external force is assumed to be the same on all species and that pressure diffusion is negligible, then $\vec{d}_i = \nabla X_i$. Since the diffusive mass flux vector is $\vec{J}_i = \rho_i \vec{V}_i$, the above equation can be written as

$$\sum_{\substack{j=1 \\ j \neq i}}^{N} \frac{X_i X_j}{\mathcal{D}_{ij}} \left(\frac{\vec{J}_j}{\rho_j} - \frac{\vec{J}_i}{\rho_i} \right) = \nabla X_i - \frac{\nabla T}{T} \sum_{\substack{j=1 \\ j \neq i}}^{N} \frac{X_i X_j}{\mathcal{D}_{ij}} \left(\frac{D_{T,j}}{\rho_j} - \frac{D_{T,i}}{\rho_i} \right) \tag{7.73}$$

After some mathematical manipulations, the diffusive mass flux vector, $\vec{J}_i$, can be obtained from

$$\vec{J}_i = -\sum_{j=1}^{N-1} \rho D_{ij} \nabla Y_j - D_{T,i} \frac{\nabla T}{T} \tag{7.74}$$

where Y_j is the mass fraction of species j. Other terms are defined as follows:

$$\begin{aligned}
D_{ij} &= [D] = [A]^{-1}[B] \\
M_{w,m} &= \left(\sum_{i=1}^{N} \frac{Y_i}{M_{w,i}} \right) \\
A_{ii} &= -\left(\frac{X_i}{\mathcal{D}_{iN}} \frac{M_{w,m}}{M_{w,N}} + \sum_{\substack{j=1 \\ j \neq i}}^{N} \frac{X_j}{\mathcal{D}_{ij}} \frac{M_{w,m}}{M_{w,i}} \right) \\
A_{ij} &= X_i \left(\frac{1}{\mathcal{D}_{ij}} \frac{M_{w,m}}{M_{w,j}} - \frac{1}{\mathcal{D}_{iN}} \frac{M_{w,m}}{M_{w,N}} \right) \\
B_{ii} &= -\left(X_i \frac{M_{w,m}}{M_{w,N}} + (1 - X_i) \frac{M_{w,m}}{M_{w,i}} \right) \\
B_{ij} &= X_i \left(\frac{M_{w,m}}{M_{w,j}} - \frac{M_{w,m}}{M_{w,N}} \right)
\end{aligned} \tag{7.75}$$

where,

$[A]$ and $[B]$ = $(N-1) \times (N-1)$ matrices

$[D]$ = an $(N-1) \times (N-1)$ matrix of the generalized Fick's law diffusion coefficients D_{ij} [120] (p. 2893)

M_w = the molecular weight

m = a subscript used to define a mean molecular weight in the mixture

7.9.3. Anisotropic Species Diffusion

You can model anisotropic species diffusion in porous media.

For Fickian diffusion, the mass flux vector of species i is modeled as,

$$J_i = -\varepsilon \left(\rho D_{i,m} + \frac{\mu_t}{Sc_t} \right) \nabla Y_i - (1 - \varepsilon) \left(\rho D_{i,m} + \frac{\mu_t}{Sc_t} \right) \mathbf{K} \nabla Y_i - D_{T,i} \frac{\nabla T}{T} \tag{7.76}$$

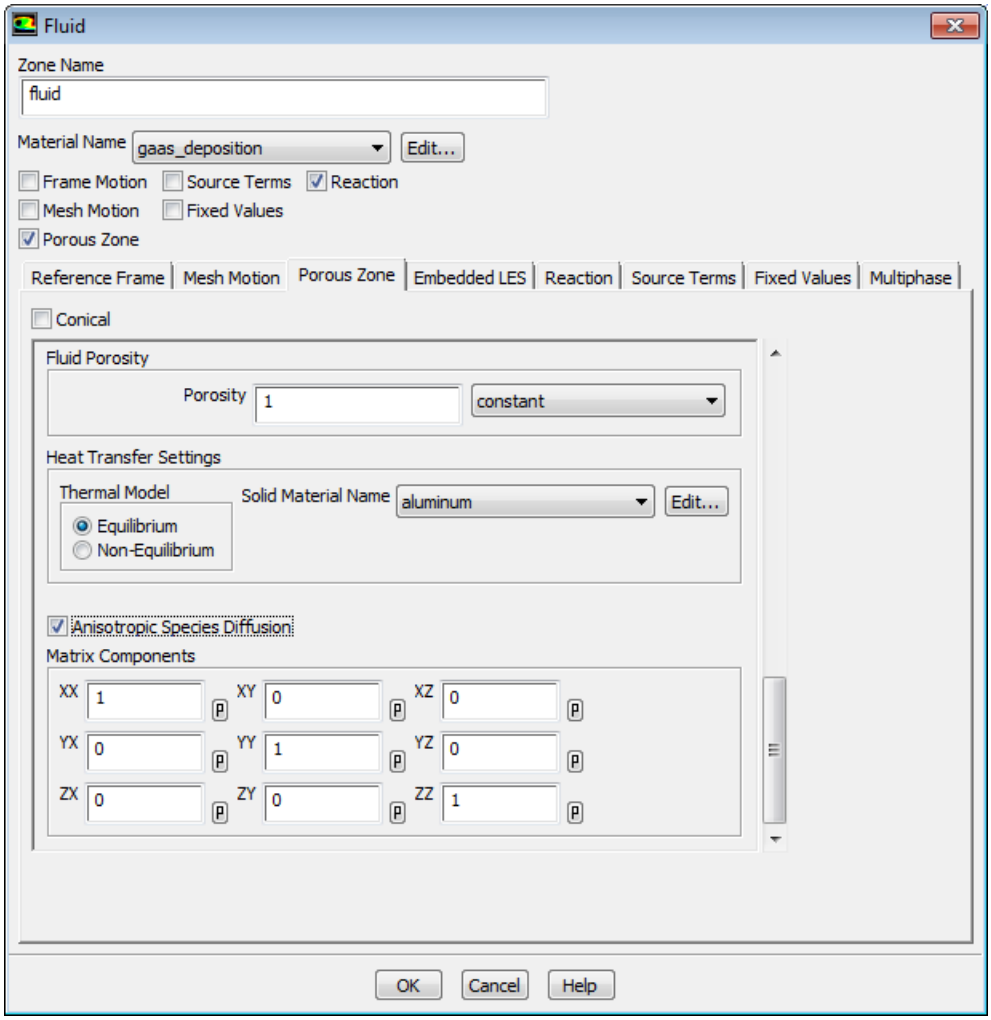
where ε is the porosity, $\mathbf{K}$ is the anisotropic diffusion matrix in the porous zone, and the remaining nomenclature is the same as in Equation 7.70 (p. 520).

The Full Multi-component (Maxwell-Stefan) anisotropic diffusion flux vector is calculated similar to Equation 7.74 (p. 521) as,

$$J_i = -\varepsilon \sum_{j=1}^{N-1} \rho D_{ij} \nabla Y_j - (1-\varepsilon) \sum_{j=1}^{N-1} \rho D_{ij} \mathbf{K} \nabla Y_j - D_{T,i} \frac{\nabla T}{T} \tag{7.77}$$

To account for anisotropic species diffusion in porous media simulations, select the **Anisotropic Species Diffusion** check box in the **Porous Zone** tab of the **Fluid** dialog box and specify the **Matrix Components** for the anisotropic diffusion matrix $\mathbf{K}$ (see Figure 7.32: Anisotropic Species Diffusion Matrix (p. 522)).

Figure 7.32: Anisotropic Species Diffusion Matrix



By default, $\mathbf{K}$ is the identity matrix that corresponds to isotropic diffusion.

7.9.4. Thermal Diffusion Coefficients

The thermal diffusion coefficients can be defined as constants, polynomial functions, user-defined functions, or using the following empirically-based composition-dependent expression derived from [55] (p. 2890):

$$D_{T,i} = -2.59 \times 10^{-7} T^{0.659} \left[\frac{M_{w,i}^{0.511} X_i}{\sum_{i=1}^{N} M_{w,i}^{0.511} X_i} - Y_i \right] \cdot \left[\frac{\sum_{i=1}^{N} M_{w,i}^{0.511} X_i}{\sum_{i=1}^{N} M_{w,i}^{0.489} X_i} \right] \tag{7.78}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This form of the Soret diffusion coefficient will cause heavy molecules to diffuse less rapidly, and light molecules to diffuse more rapidly, towards heated surfaces.

7.9.4.1. Thermal Diffusion Coefficient Inputs

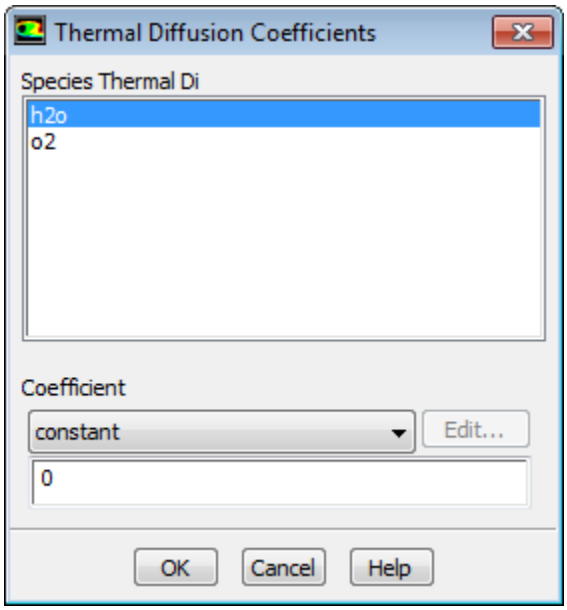
If you have enabled thermal diffusion (in the Species Model Dialog Box (p. 2216)), you can define the thermal diffusion coefficients in the Create/Edit Materials Dialog Box (p. 2302) as follows:

1. Select one of the following three methods in the drop-down list to the right of **Thermal Diffusion Coefficient**:

   - Choose **kinetic-theory** to have ANSYS Fluent compute the thermal diffusion coefficients using the empirically-based expression in Equation 7.78 (p. 522). No further inputs are required for this option.
   - Choose **specified** to input the coefficient for each species. The Thermal Diffusion Coefficients Dialog Box (p. 2346) (Figure 7.33: The Thermal Diffusion Coefficients Dialog Box (p. 523)) will open. Further inputs are described in the step 2.
   - Choose **user-defined** to use a user-defined function. More information about user-defined functions can be found in the Fluent Customization Manual.

2. (**specified** method only) For each species in the **Species Thermal Di** list of the **Thermal Diffusion Coefficients** dialog box, perform the following steps:

   Figure 7.33: The Thermal Diffusion Coefficients Dialog Box

   a. Select the species in the **Species Thermal Di** list for which you are going to define the thermal diffusion coefficient.

   b. Define $D_{T,i}$ for the selected species either as a constant value or as a polynomial function of temperature:

      - To define a constant diffusion coefficient, select **constant** (the default) in the drop-down list below **Coefficient**, and then enter the value in the field below the list.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To define a temperature-dependent diffusion coefficient, choose **polynomial** in the **Coefficient** drop-down list and then define the polynomial coefficients as described in Defining Properties Using Temperature-Dependent Functions (p. 475).

7.9.5. Mass Diffusion Coefficient Inputs

By default, the solver computes the species diffusion using Equation 7.68 (p. 519) (for laminar flows) with your inputs for $D_{i,m}$, the diffusion coefficient for species i in the mixture. For turbulent flows, species diffusion is computed with Equation 7.70 (p. 520).

You can input the mass diffusion coefficients using one of the following methods:

- Constant dilute approximation (Fickian diffusion only): define one constant for all $D_{i,m}$.
- Dilute approximation (Fickian diffusion only): define each $D_{i,m}$ as a constant or as a polynomial function of temperature (if heat transfer is enabled).
- Multicomponent method: define the binary diffusion of species i in each species j, $\mathcal{D}_{ij}$ as a constant or a polynomial function of temperature, or (for ideal gases only) using kinetic theory.
- Unity Lewis Number: define $D_{i,m}$ under the assumption of unity Lewis number for all species in the mixture.
- User-defined function (UDF): define a single function that will apply to all mass diffusion coefficients. This is done using the `DEFINE_DIFFUSIVITY` macro and is explained in the Fluent Customization Manual.

You should choose to input $D_{i,m}$ (using one of the first two methods) if you are modeling a dilute mixture, with chemical species present at low mass fraction in a "carrier" fluid that is present at high concentration. You may want to define the individual binary mass diffusion coefficients, $\mathcal{D}_{ij}$, if you are modeling a non-dilute mixture. If you choose to define $\mathcal{D}_{ij}$, the solver will compute the diffusion of species i in the mixture using Equation 7.69 (p. 519), unless you have enabled full multicomponent diffusion.

The Lewis number is the ratio of thermal diffusivity (Prandtl number) to mass diffusivity (Schmidt number). Flames with non-unity Lewis number can display instabilities, as well as non-adiabatic flame temperatures. The Unity Lewis Number mass diffusivity option avoids these issues, and is particularly appropriate for the Thickened Flame model.

Important

If you want to use the full multicomponent diffusion model described in Full Multicomponent Diffusion (p. 520), turn on the **Full Multicomponent Diffusion** option in the Species Model Dialog Box (p. 2216), and then select the multicomponent diffusion model.

You will define $D_{i,m}$ or $\mathcal{D}_{ij}$ for each chemical species using the Create/Edit Materials Dialog Box (p. 2302).

Setup → Materials

The diffusion coefficients have units of m^2/s in SI units or ft^2/s in British units.

7.9.5.1. Constant Dilute Approximation Inputs

To use the constant dilute approximation method, follow these steps:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select **constant-dilute-appx** in the drop-down list to the right of **Mass Diffusivity**.

2. Enter a single value of $D_{i,m}$. The same value will be used for the diffusion coefficient of each species in the mixture.

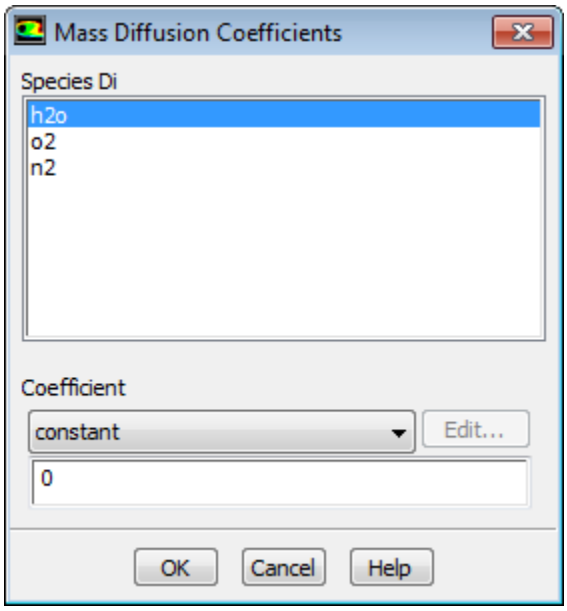
7.9.5.2. Dilute Approximation Inputs

To use the dilute approximation method, follow the steps below:

1. Select **dilute-approx** in the drop-down list to the right of **Mass Diffusivity**.

2. In the resulting Mass Diffusion Coefficients Dialog Box (p. 2344) (Figure 7.34: The Mass Diffusion Coefficients Dialog Box for Dilute Approximation (p. 525)), select the species in the **Species Di** list for which you are going to define the mass diffusion coefficient.

   Figure 7.34: The Mass Diffusion Coefficients Dialog Box for Dilute Approximation



3. You can define $D_{i,m}$ for the selected species either as a constant value or (if heat transfer is active) as a polynomial function of temperature:

   - To define a constant diffusion coefficient, select **constant** (the default) in the drop-down list below **Coefficient**, and then enter the value in the field below the list.

   - To define a temperature-dependent diffusion coefficient, choose **polynomial** in the **Coefficient** drop-down list and then define the polynomial coefficients as described in Inputs for Polynomial Functions (p. 475).

$$D_{i,m} = A_1 + A_2 T + A_3 T^2 + \ldots \tag{7.79}$$

4. Repeat steps 2 and 3 until you have defined diffusion coefficients for all species in the **Species Di** list in the Mass Diffusion Coefficients Dialog Box (p. 2344).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

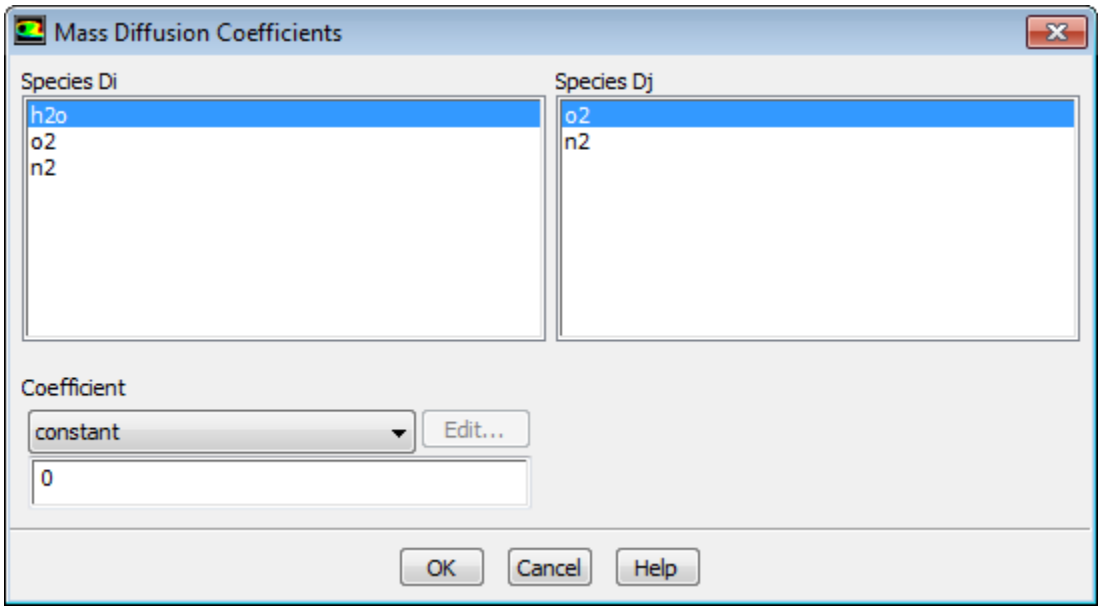
7.9.5.3. Multicomponent Method Inputs

To use the multicomponent method, and define constant or temperature-dependent diffusion coefficients, follow the steps below:

1. Select **multicomponent** in the drop-down list to the right of **Mass Diffusivity**.

2. In the resulting Mass Diffusion Coefficients Dialog Box (p. 2344) (Figure 7.35: The Mass Diffusion Coefficients Dialog Box for the Multicomponent Method (p. 526)), select the species in the **Species Di** list and the **Species Dj** list for which you are going to define the mass diffusion coefficient D_{ij} for species i in species j.

   Figure 7.35: The Mass Diffusion Coefficients Dialog Box for the Multicomponent Method



3. You can define $\mathcal{D}_{ij}$ for the selected pair of species as a constant value or as a polynomial function of temperature (if heat transfer is active).

   - To define a constant diffusion coefficient, select **constant** (the default) in the drop-down list below **Coefficient**, and then enter the value in the field below the list.

   - To define a temperature-dependent diffusion coefficient, choose **polynomial** in the **Coefficient** drop-down list and then define the polynomial coefficients as described in Inputs for Polynomial Functions (p. 475).

     $$\mathcal{D}_{ij} = A_1 + A_2 T + A_3 T^2 + \ldots \tag{7.80}$$

4. Repeat steps 2 and 3 until you have defined diffusion coefficients for all pairs of species in the **Species Di** and **Species Dj** lists in the Mass Diffusion Coefficients Dialog Box (p. 2344).

To use the multicomponent method, and define the diffusion coefficient using kinetic theory (available only when the ideal gas law is used), follow these steps:

1. Choose **kinetic-theory** in the drop-down list to the right of **Mass Diffusivity**.

2. Click **Change/Create** after completing other property definitions for the mixture material.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Define the Lennard-Jones parameters, σ_i and $\left(\varepsilon / k_B\right)_i$, for each species (fluid material), as described in Kinetic Theory Parameters (p. 529).

The solver will use a modification of the Chapman-Enskog formula [67] (p. 2890) to compute the diffusion coefficient using kinetic theory:

$$\mathcal{D}_{ij}=0.00188\frac{\left[T^3\left(\frac{1}{M_{w,i}}+\frac{1}{M_{w,j}}\right)\right]^{1/2}}{p_{abs}\sigma_{ij}^2\Omega_D} \tag{7.81}$$

where,

p_{abs} is the absolute pressure

Ω_D is the diffusion collision integral, which is a measure of the interaction of the molecules in the system

Ω_D is a function of the quantity T_D^*, where

$$T_D^*=\frac{T}{\left(\varepsilon / k_B\right)_{ij}} \tag{7.82}$$

k_B is the Boltzmann constant, which is defined as the gas constant, R, divided by Avogadro's number. $\left(\varepsilon / k_B\right)_{ij}$ for the mixture is the *geometric* average:

$$\left(\varepsilon / k_B\right)_{ij}=\sqrt{\left(\varepsilon / k_B\right)_i\left(\varepsilon / k_B\right)_j} \tag{7.83}$$

For a binary mixture, σ_{ij} is calculated as the *arithmetic* average of the individual σs:

$$\sigma_{ij}=\frac{1}{2}\left(\sigma_i+\sigma_j\right) \tag{7.84}$$

7.9.5.4. Unity Lewis Number

The laminar mass diffusivity of all species is calculated as:

$$D_{i,m}=\frac{k}{\rho c_p}$$

where $D_{i,m}$ denotes the mass diffusivity of species i in the mixture, k is the thermal conductivity, ρ is the mixture density, and c_p is the mixture specific heat.

7.9.6. Mass Diffusion Coefficient Inputs for Turbulent Flow

When your flow is turbulent, you will define $D_{i,m}$ or $\mathcal{D}_{ij}$, as described for laminar flows in Mass Diffusion Coefficient Inputs (p. 524), and you will also have the option to alter the default setting for the turbulent Schmidt number, Sc_t, as defined in Equation 7.71 (p. 520).

Usually, in a turbulent flow, the mass diffusion is dominated by the turbulent transport as determined by the turbulent Schmidt number (Equation 7.71 (p. 520)). The turbulent Schmidt number measures the relative diffusion of momentum and mass due to turbulence and is on the order of unity in all turbulent flows. Because the turbulent Schmidt number is an empirical constant that is relatively insensitive to the molecular fluid properties, you will have little reason to alter the default value (0.7) for any species.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Should you want to modify the Schmidt number, enter a new value for **Turb. Schmidt Number** in the Viscous Model Dialog Box (p. 2176).

Setup → Models → Viscous Edit...

Important

Note that the full multicomponent diffusion model described in Full Multicomponent Diffusion (p. 520) is not recommended for turbulent flows.

7.10. Standard State Enthalpies

When you are solving a reacting flow using the finite-rate or eddy dissipation model, you must define the standard state enthalpy (also known as the formation enthalpy or heat of formation), h_j^0 for each species j. These inputs are used to define the mixture enthalpy as

$$H = \sum_j Y_j \left[\frac{h_j^0}{M_j} + \int_T^{T_{ref,j}} c_{p,j} dT \right] \tag{7.85}$$

where M_j is the molecular weight of the j^{th} species with units of kg/kmol and $T_{ref,j}$ is the reference temperature at which h_j^0 is defined. Standard state enthalpies are input in units of J/kg mol in SI units or in units of Btu/lb_mmol in British units.

For each species involved in the reaction (that is, each fluid material contained in the mixture material), you can set the **Standard State Enthalpy** and **Reference Temperature** in the Create/Edit Materials Dialog Box (p. 2302).

7.11. Standard State Entropies

If you are using the finite-rate model with reversible reactions (see Direct Use of Finite-Rate Kinetics (no TCI) in the Theory Guide), you must define the standard state entropy, s_j^0 for each species j. These inputs are used to define the mixture entropy as

$$S = \sum_j Y_j \left[\frac{s_j^0}{M_j} + \int_T^{T_{ref,j}} \frac{c_{p,j}}{T} dT \right] \tag{7.86}$$

where M_j is the molecular weight of the j^{th} species with units of kg/kmol and $T_{ref,j}$ is the reference temperature at which s_j^0 is defined. Standard state entropies are input in units of J/kmol-K in SI units or in units of Btu/lb_mmol-°R in British units.

For each species involved in the reaction (that is, each fluid material contained in the mixture material), you can set the **Standard State Entropy** and **Reference Temperature** in the Create/Edit Materials Dialog Box (p. 2302).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.12. Unburnt Thermal Diffusivity

If you are modeling premixed combustion (see Modeling Premixed Combustion (p. 1089)), the fluid material in your domain should be assigned the properties of the unburnt mixture, including the thermal diffusivity (α in Equation 9.9 in the Theory Guide). α is defined as $k/\rho c_p$, and values at standard conditions can be found in combustion handbooks (for example, [55] (p. 2890)). To determine values at non-standard conditions, you must use a third-party 1D combustion program with detailed chemistry. You can set the **Unburnt Thermal Diffusivity** in the Create/Edit Materials Dialog Box (p. 2302).

7.13. Kinetic Theory Parameters

You may choose to define the following properties using kinetic theory when the ideal gas law is enabled:

- viscosity (for fluids)
- thermal conductivity (for fluids)
- specific heat capacity (for fluids)
- mass diffusion coefficients (for multi-species mixtures)

If you are using kinetic theory for a fluid's viscosity (Equation 7.27 (p. 491)), you must input the kinetic theory parameters σ and ε/k_B for that fluid. These parameters are the Lennard-Jones parameters and are referred to by ANSYS Fluent as the "characteristic length" and the "energy parameter" respectively.

- When kinetic theory is applied to calculation of a fluid's thermal conductivity only, no inputs are required.
- To calculate specific heat of a fluid using kinetic theory (Equation 7.65 (p. 515)), you need to enter the degrees of freedom for the fluid material.
- If you use kinetic theory to define a mixture material's mass diffusivity (Equation 7.81 (p. 527)), you must input σ_i and $\left(\varepsilon/k_B\right)_i$ for each chemical species i.

For additional information, see the following section:
7.13.1. Inputs for Kinetic Theory

7.13.1. Inputs for Kinetic Theory

The procedure for using kinetic theory is as follows:

1. Select **kinetic-theory** as the property specification method for the **Viscosity**, **Thermal Conductivity**, or heat capacity **Cp** of a fluid material, or for the **Mass Diffusivity** of a mixture material.
2. If the material for which you have selected the kinetic theory method for one or more properties is a fluid material, you must set the kinetic theory parameters for each of the constituent species (fluid materials).

   The parameters to be set are as follows:

   - **L-J Characteristic Length**
   - **L-J Energy Parameter**
   - **Degrees of Freedom** (only required if kinetic theory is used for specific heat)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See the beginning of this section to find out which parameters are required to calculate each property using kinetic theory.

Characteristic length is defined in units of Angstroms. The energy parameter is defined in units of absolute temperature. Degrees of freedom is a dimensionless input. All kinetic theory materials can be found in the literature (for example, [44] (p. 2889)).

7.14. Operating Pressure

Specification of the operating pressure affects your calculation in different ways for different flow regimes. This section presents information about the operating pressure, its relevance for different cases, and how to set it correctly.

For additional information, see the following sections:

7.14.1. The Significance of Operating Pressure
7.14.2. Operating Pressure, Gauge Pressure, and Absolute Pressure
7.14.3. Setting the Operating Pressure

7.14.1. The Significance of Operating Pressure

Operating pressure is significant for incompressible ideal gas flows because it directly determines the density: the incompressible ideal gas law computes density as $\rho=\frac{p_{op}}{\frac{R}{M_w}T}$. You must therefore be sure to set the operating pressure appropriately.

In low-Mach-number compressible flow, the overall pressure drop is small compared to the absolute static pressure, and can be significantly affected by numerical roundoff. To understand why this is true, consider a compressible flow with $M<<1$. The pressure changes, Δp, are related to the dynamic head, $\frac{1}{2}\gamma pM^2$, where p is the static pressure and γ is the ratio of specific heats. This gives the simple relationship $\Delta p/\ p\sim M^2$, so that $\Delta p/\ p\rightarrow 0$ as $M\rightarrow 0$. Therefore, unless adequate precaution is taken, low-Mach-number flow calculations are very susceptible to roundoff error.

Operating pressure is significant for low-Mach-number compressible flows because of its role in avoiding roundoff error problems. Ensure that you set the operating pressure appropriately. You may want to specify a floating operating pressure instead of a constant operating pressure for low-Mach-number, time-dependent compressible flows with average pressure in the domain varying in time. See Floating Operating Pressure (p. 597) for details.

Operating pressure is less significant for higher-Mach-number compressible flows. The pressure changes in such flows are much larger than those in low-Mach-number compressible flows, so there is no real problem with roundoff error and, therefore, no real need to use gauge pressure. In fact, it is common convention to use absolute pressures in such calculations. Since ANSYS Fluent always uses gauge pressure, you can simply set the operating pressure to zero, making gauge and absolute pressures equivalent.

If the density is assumed constant or if it is derived from a profile function of temperature, the operating pressure is not used in the density calculation.

Note that the default operating pressure is 101325 Pa.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.14.2. Operating Pressure, Gauge Pressure, and Absolute Pressure

ANSYS Fluent avoids the problem of roundoff error (discussed in The Significance of Operating Pressure (p. 530)) by subtracting the operating pressure (generally a large pressure roughly equal to the average absolute pressure in the flow) from the absolute pressure, and using the result (termed the gauge pressure). The relationship between the operating pressure, gauge pressure, and absolute pressure is shown below. The absolute pressure is simply the sum of the operating pressure and the gauge pressure:

$$p_{abs} = p_{op} + p_{gauge} \tag{7.87}$$

All pressures that you specify and all pressures computed or reported by ANSYS Fluent are gauge pressures.

7.14.3. Setting the Operating Pressure

The criteria for choosing a suitable operating pressure are based on the Mach-number regime of the flow and the relationship that is used to determine density. For example, if you use the ideal gas law in an incompressible flow calculation (for example, for a natural convection problem), you should use a value representative of the mean flow pressure.

To place this discussion in perspective, Table 7.1: Recommended Settings for Operating Pressure (p. 531) shows the recommended approach for setting operating pressures. The default operating pressure is 101325 Pa.

Table 7.1: Recommended Settings for Operating Pressure

Density Relationship	Mach Number Regime	Operating Pressure
ideal gas law	$M > 0.1$	0 or ≈ mean flow pressure
ideal gas law	$M < 0.1$	≈ mean flow pressure
profile function of temperature	incompressible	not used
constant	incompressible	not used
incompressible ideal gas law	incompressible	≈ mean flow pressure

You will set the **Operating Pressure** in the Operating Conditions Dialog Box (p. 2382).

Setup → **Cell Zone Conditions** → **Operating Conditions...**

7.15. Reference Pressure Location

For incompressible flows that do not involve any pressure boundaries, ANSYS Fluent adjusts the gauge pressure field after each iteration to keep it from floating. This is done using the pressure in the cell located at (or nearest to) the reference pressure location. The pressure value in this cell is subtracted from the entire gauge pressure field; as a result, the gauge pressure at the reference pressure location is always zero. If pressure boundaries are involved, the adjustment is not needed and the reference pressure location is ignored.

The reference pressure location is, by default, the cell center at or closest to (0,0,0). There may be cases in which you might want to move the reference pressure location, perhaps locating it at a point where the absolute static pressure is known (for example, if you are planning to compare your results with experimental data). To change the location, enter new (**X,Y,Z**) coordinates for **Reference Pressure Location** in the Operating Conditions Dialog Box (p. 2382).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → Cell Zone Conditions → Operating Conditions...

For additional information, see the following section:
7.15.1. Actual Reference Pressure Location

7.15.1. Actual Reference Pressure Location

For cases that do not have pressure-related boundary conditions (for example, pressure inlet, pressure outlet, pressure far-field, and so on), you need to specify the **Reference Pressure Location** at a point in the problem domain. Internally, ANSYS Fluent sets the location of the reference pressure at a slightly different nearby location. Therefore, the actual location used as the pressure reference is different than that of your input value. To report the actual reference pressure location that ANSYS Fluent uses, use the following text command:

`define → operating-conditions → used-ref-pressure-location`

Important

- This text command is available only when the case is initialized and has no pressure-related boundary zones.
- Reporting the actual reference pressure location is not available through the graphical user interface.

7.16. Real Gas Models

Some engineering problems involve fluids that do not behave as ideal gases. For example, at very high-pressure or very low-temperature conditions (for example, the flow of a refrigerant through a compressor) the flow cannot typically be modeled accurately using the ideal-gas assumption. Therefore, the real gas model allows you to solve accurately for the fluid flow and heat transfer problems where the working fluid behavior deviates from the ideal-gas assumption.

ANSYS Fluent provides three real gas options for solving these types of flows:

- Cubic Equation of State Models (p. 535)
- The NIST Real Gas Models (p. 550)
- The User-Defined Real Gas Model (p. 561)

All the models allow you to solve for either a single-species fluid flow or a multiple-species mixture fluid flow.

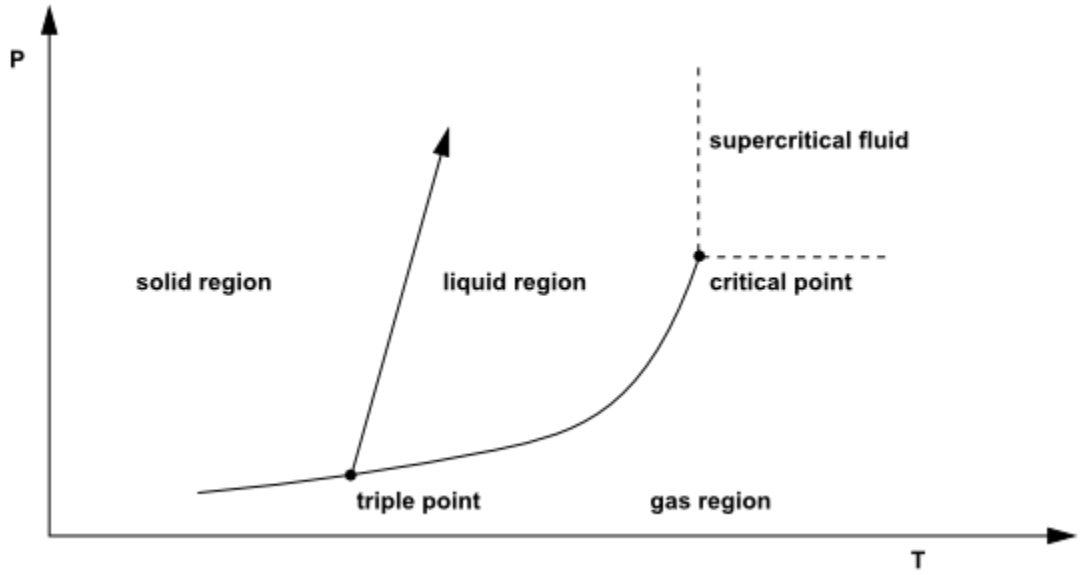
For additional information, see the following sections:
7.16.1. Introduction
7.16.2. Choosing a Real Gas Model
7.16.3. Cubic Equation of State Models
7.16.4. The NIST Real Gas Models
7.16.5. The User-Defined Real Gas Model

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.16.1. Introduction

The states at which a pure material can exist can be graphically represented in diagrams of pressure vs. temperature (PT diagrams) and pressure vs. molecular or specific volume (PV diagrams). Homogeneous fluids are normally divided into two classes, liquids and gases. However the distinction cannot always be sharply drawn, because the two phases become indistinguishable at what is called the critical point. A typical pressure-temperature (PT) diagram of a pure material is shown in Figure 7.36: Typical PT Diagram of a Pure Material (p. 533).

Figure 7.36: Typical PT Diagram of a Pure Material



This figure shows the single phase regions, as well as the conditions of P and T where two phases coexist. Therefore the solid and the gas region are divided by the sublimation curve, the liquid and gas regions by the vaporization curve, and the solid and liquid regions by the fusion curve. The three curves meet at the triple point, where all three phases coexist in equilibrium. Although the fusion curve continues upward indefinitely, the vaporization curve terminates at the critical point. The coordinates of this point are called the critical pressure P_c and critical temperature T_c. These represent the highest temperature and pressure at which a pure material can exist in vapor-liquid equilibrium. At temperatures and pressures above the critical point, the physical property differences that differentiate the liquid phase from the gas phase become less defined. This reflects the fact that, at extremely high temperatures and pressures, the liquid and gaseous phases become indistinguishable. This new phase, which has some properties that are similar to a liquid and some properties that are similar to a gas, is called a supercritical fluid.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.37: Typical PV Diagram of a Pure Material

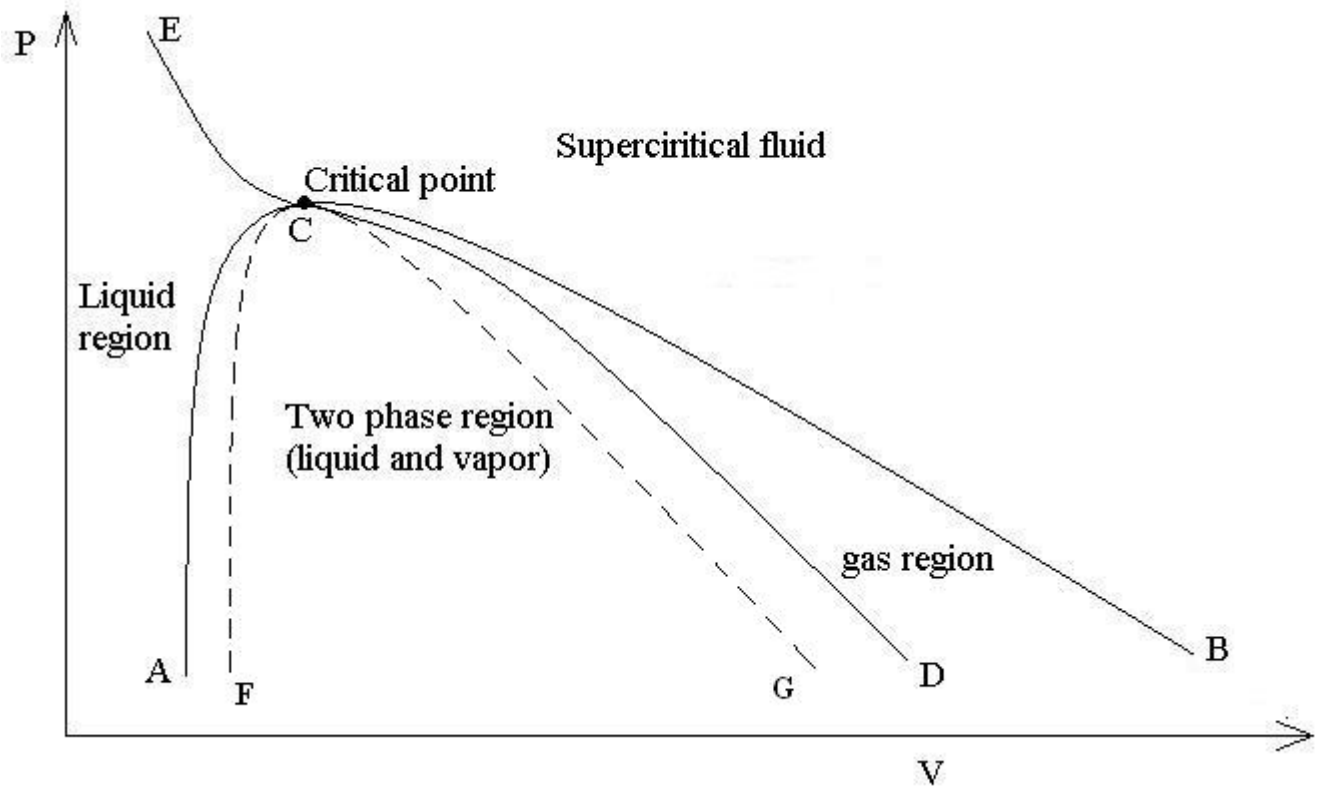


Figure 7.37: Typical PV Diagram of a Pure Material (p. 534) presents a typical diagram of pressure versus molar or specific volume (PV diagram) of a pure material. The dome shaped curve ACD is called the saturation dome and separates the single phase regions in the diagram; curve AC represents the saturated liquid and curve CD the saturated vapor. The area under the saturation dome ACD is the two-phase region and represents all possible mixtures of vapor and liquid in equilibrium. Curve ECB is the critical isotherm and exhibits a horizontal inflection at point C at the top of the dome. This is the critical point. The specific volume corresponding to the critical point, is called the critical specific volume V_c. The conditions to the right of the critical isotherm ECB correspond to supercritical fluid. The dashed lines CF and CG in Figure 7.37: Typical PV Diagram of a Pure Material (p. 534) represent the liquid and the vapor spinodal curves with the regions ACF and DCG between the saturation and spinodal lines representing the superheated liquid and supercooled vapor states, respectively. These states are called "metastable" because they exist temporarily in small local regions until phase change occurs. As an example, liquid metastable states may be formed in situations of fast depressurization, depending on the rate of depressurization and the disturbances that tend to make the liquid flash into vapor.

7.16.2. Choosing a Real Gas Model

The equation of state is the mathematical expression that relates pressure, molar or specific volume, and temperature for any pure homogeneous fluid in equilibrium states.

The simplest equation of state is the ideal gas law, which is approximately valid for the low pressure gas region of the PT and PV diagrams. Ideal gas behavior can be expected when

$$P/P_c \ll 1$$

or

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$T/T_c > 2$ and $P/P_c < 1$

If your flow conditions correspond to either of those cases, you may use the ideal gas law in your simulation.

Another idealization, that of the incompressible fluid, can be employed for the low pressure region of the liquid phase. A constant density option is the appropriate selection in that case.

However, both of these approaches are not good approximations for flow conditions close to and beyond the critical point, where the fluid behavior cannot be described by the ideal gas, or the incompressible liquid assumptions. We refer to a fluid under those conditions as a real fluid, or a real gas and more complex relations are used for the determination of its physical and thermodynamic properties.

ANSYS Fluent provides the following options for solving real fluid problems:

- The cubic equation of state models can be used to solve problems in the gas, liquid, and supercritical fluid regimes. The models are not available for the two-phase region under the phase dome. For further details see Cubic Equation of State Models (p. 535).
- The NIST real gas model can be used to solve problems in the liquid, or gas and supercritical fluid regimes. The model does not allow modeling of the two-phase region. For further details see The NIST Real Gas Models (p. 550).
- The user-defined real gas model allows you to solve problems in all regimes, as long as appropriate relationships are provided through the user-defined real gas functions. For further details see The User-Defined Real Gas Model (p. 561).

The concepts presented in this section for pure materials are also extended to multicomponent mixtures with the introduction of appropriate composition-dependent parameters in the real gas equations of state and the material property models. All the real-gas modeling options above allow for either single-species or multicomponent flow modeling. In addition, you may solve reacting flow problems with the cubic equations of state models and the user-defined real gas functions.

7.16.3. Cubic Equation of State Models

7.16.3.1. Overview and Limitations

An equation of state is a thermodynamic equation, which provides a mathematical relationship between two or more state functions associated with the matter, such as its temperature, pressure, volume, or internal energy. One of the simplest equations of state for this purpose is the ideal gas law, which is roughly accurate for gases at low pressures and high temperatures. However, this equation becomes increasingly inaccurate at higher pressures and lower temperatures, and fails to predict condensation from a gas to a liquid.

Introduced in 1949, the Redlich-Kwong equation of state [87] (p. 2891) was a considerable improvement over other equations of that time. It is an analytic cubic equation of state and is still of interest primarily due to its relatively simple form. The original form is

$$P = \frac{RT}{V-b} - \frac{\alpha_0}{V(V+b)T_r^{0.5}} \tag{7.88}$$

where

P = absolute pressure (Pa)

R = universal gas constant

V = specific molar volume ($\mathrm{m}^3/\mathrm{kmol}$)

T = temperature (K)

T_r = reduced temperature $\frac{T}{T_c}$, where T_c is the critical temperature

α_0 and b are constants related directly to the fluid critical pressure and temperature

Many investigators have attempted to improve the accuracy of the Redlich-Kwong equation. ANSYS Fluent has adopted the original form of the Redlich-Kwong equation, as well as the following modified forms:

- The Soave-Redlich-Kwong [112] (p. 2893) equation is a three-parameter equation of state, which can be applied for vapor, supercritical, and liquid property predictions. It has found wide acceptance mainly in the oil and gas industry and requires knowledge of the critical temperature, critical pressure, and acentric factor. It was developed by replacing the Redlich-Kwong attractive coefficient defined as $\alpha=\alpha_0/T_r^{0.5}$ with a two-parameter form $\alpha(T_r,\omega)$, where ω is the acentric factor.

- The Peng-Robinson equation [82] (p. 2891) is a three-parameter equation of state, also requiring the critical temperature, critical pressure and acentric factor parameters. It is thought to perform as well as the Soave-Redlich-Kwong equation, with an advantage in the prediction of the liquid densities.

- The Aungier-Redlich-Kwong [8] (p. 2887) equation provides improved predictions for vapor and supercritical fluids near the critical point, as well as for materials with a negative value of the acentric factor. The Aungier modified form is a four parameter equation and requires the critical specific volume in addition to the critical temperature, critical pressure and acentric factor.

The following limitations exist in ANSYS Fluent for all cubic equation of state models:

- Pressure-inlets, mass flow-inlets, and pressure-outlets are the only inflow and outflow boundaries available for use with the real gas models.

- Non-reflecting boundary conditions are not compatible with the real gas models when using the density-based solver. If your model requires NRBC and a real gas model, you must use the pressure-based solver.

- The cubic equation of state real gas models are compatible with the Eulerian multiphase models.

Note

Note the following restrictions:

– Only one phase can be a real gas.

- The cubic equation of state models are compatible with the Lagrangian Dispersed Phase Models. If you are modeling droplet or multicomponent particles, note that the current formulation does not take into account the near-critical point phenomena, which means that accurate results can be obtained for droplet temperatures below $T_{\lim}=0.9T_c$. See Using the Cubic Equation of State Models with the Lagrangian Dispersed Phase Models (p. 548) for more details.

- The real gas models are not available with the premixed, inert, and composition PDF transport combustion models.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The cubic equation of state real gas models can be used with the following species models:
  - Species transport. Chemical reactions can be modeled with the finite rate and eddy dissipation models. Note that the **Dimension Reduction** model is not available with the real gas models.
  - Non-premixed model and partially-premixed model. Note that the **Compressibility Effects** option must be enabled in the **Species Model** dialog box in order for the real-gas models to be available. In addition the following restrictions apply when the non-premixed model is used together with a cubic equation of state real-gas model:
    → The empirical stream options cannot be used, as for the empirical species the critical properties cannot be defined.
    → Condensed species such as h2o<l> and c<s> are not supported and should be excluded from the PDF table, so ensure that you add all condensed species for your system in the excluded species list prior to the PDF table generation.
    The laminar flame speeds for the partially-premixed model are assumed to be the same as for ideal-gases.
- When the cubic equation of state models are applied for subcritical conditions near or under the phase dome, the two-phase flow is not modeled and either vapor or liquid state must be selected. Subcritical and supercritical states can co-exist in the same simulation.

7.16.3.2. Equation of State

The general form of pressure P for the cubic equation of state models is written as [8] (p. 2887) :

$$P=\frac{RT}{V-b+c}-\frac{\alpha}{\left(V^2+\delta V+\varepsilon\right)} \tag{7.89}$$

where

P = absolute pressure (Pa)

V = specific molar volume ($\mathrm{m}^3/\mathrm{kmol}$)

T = temperature (K)

R = universal gas constant

The coefficients α, b, c, δ, and ε are given for each equation of state as functions of the critical temperature T_c, critical pressure P_c, acentric factor ω and critical specific volume V_c. Note that the attractive coefficient α also has a temperature dependence, which varies for each equation of state model, and is commonly written as $\alpha=\alpha(T)$.

Redlich-Kwong Equation [92] (p. 2892):

$$\alpha(T)=\frac{\alpha_0}{\left(T/T_c\right)^{0.5}} \tag{7.90}$$

$$\alpha_0=\frac{0.42747R^2T_c^2}{P_c} \tag{7.91}$$

$$b=\frac{0.08664RT_c}{P_c} \tag{7.92}$$

The parameter δ is set equal to b, while c and ε are set to 0. The Redlich-Kwong equation is the simplest of the cubic equations of state in ANSYS Fluent and requires two parameters only, the critical temperature T_c and the critical pressure P_c.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Soave-Redlich-Kwong Equation [112] (p. 2893):

$$\alpha(T)=\alpha_0[1+n(1-(T/T_c)^{0.5})]^2 \tag{7.93}$$

$$n=0.48+1.574\omega-0.176\omega^2 \tag{7.94}$$

α_0 and b are given by Equation 7.91 (p. 537) and Equation 7.92 (p. 537) respectively. As in the original Redlich-Kwong equation the parameter δ is set equal to b, while c and ε are set to 0. The Soave-Redlich-Kwong requires three parameters, the critical temperature T_c, the critical pressure P_c, and the acentric factor ω.

Peng-Robinson Equation [82] (p. 2891):

$$\alpha_0=\frac{0.457247R^2T_c^2}{P_c} \tag{7.95}$$

$$b=\frac{0.07780RT_c}{P_c} \tag{7.96}$$

The function $\alpha(T)$ is given by Equation 7.93 (p. 538), with n provided in Equation 7.97 (p. 538) as follows:

$$n=0.37464+1.54226\omega-0.26992\omega^2 \tag{7.97}$$

In the Peng-Robinson equation δ is set equal to 2b, ε is equal to $-b^2$, and c is set to 0.

Similar to the Soave-Redlich-Kwong equation, the Peng-Robinson equation is a three-parameter equation and requires the critical temperature T_c, the critical pressure P_c, and the acentric factor ω.

Aungier-Redlich-Kwong Equation [8] (p. 2887):

$$\alpha(T)=\alpha_0\left(\frac{T}{T_c}\right)^{-n} \tag{7.98}$$

$$n=0.4986+1.1735\omega+0.4754\omega^2 \tag{7.99}$$

α_0 and b are given by Equation 7.91 (p. 537) and Equation 7.92 (p. 537), respectively. As in the original Redlich-Kwong equation, the parameter δ is set equal to b and ε is set to 0. Parameter c is given by:

$$c=\frac{RT_c}{P_c+\frac{\alpha_0}{V_c(V_c+b)}}+b-V_c \tag{7.100}$$

The Aungier-Redlich-Kwong equation requires four parameters, namely the critical temperature T_c, the critical pressure P_c, the critical specific volume V_c, and the acentric factor ω.

7.16.3.3. Enthalpy, Entropy, and Specific Heat Calculations

Enthalpy, entropy, and specific heat are computed in terms of the relevant ideal gas properties and the departure functions. The departure function F_{dep} of any conceptual property F is defined as [87] (p. 2891)

$$F_{dep}=F_{ideal}(T,P)-F(T,P) \tag{7.101}$$

where F_{ideal} is the value of the property as computed from the ideal gas relations. The departure function F_{dep} can be derived from basic thermodynamic relations and the equation of state.

Following the above definition, the enthalpy H for the equations of state models is given by the following equations [8] (p. 2887):

$$H=H_{ideal}-\frac{H_{dep}}{MW} \tag{7.102}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$H_{dep} = -PV + RT - \frac{T\frac{\partial a}{\partial T} - a}{\Delta} ln\left[\frac{2V + \delta + \Delta}{2V + \delta - \Delta}\right] \tag{7.103}$$

$$\Delta = \left(\delta^2 - 4\varepsilon\right)^{0.5} \tag{7.104}$$

where,

H_{ideal} = ideal gas enthalpy at temperature T (J/kg)

H_{dep} = departure enthalpy (J/kmol)

MW = mean molecular weight (Kg/kmol)

P = pressure (Pa)

V = specific molar volume (m^3/kmol)

R = universal gas constant

α, δ, and ε are computed using Equation 7.90 (p. 537)– Equation 7.100 (p. 538) depending on the equation of state model

See Equation 7.89 (p. 537) for a description of other coefficients.

Similarly, the departure internal energy U_{dep} can be shown to be

$$U_{dep} = -\frac{T\frac{\partial a}{\partial T} - a}{\Delta} ln\left[\frac{2V + \delta + \Delta}{2V + \delta - \Delta}\right] \tag{7.105}$$

where

U_{dep} = departure internal energy (J/kmol)

T = temperature (K)

V = specific molar volume (m^3/kmol)

Δ is given by Equation 7.104 (p. 539)

α, δ, and ε are computed using Equation 7.90 (p. 537) – Equation 7.100 (p. 538) depending on the equation of state model

The specific heat c_p can be computed from the ideal specific heat $c_{p,ideal}$ and the departure specific heat at constant volume $c_{v,dep}$ as follows:

$$c_p = c_{p,ideal} - \frac{c_{p,dep}}{MW} \tag{7.106}$$

$$c_{p,dep} = c_{v,dep} - R - T\frac{\left(\partial V / \partial T\right)^2}{\partial V / \partial P} \tag{7.107}$$

where

MW = mean molecular weight (kg/kmol)

P = pressure (Pa)

T = temperature (K)

V = specific molar volume (m^3/kmol)

R = universal gas constant

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In Equation 7.107 (p. 539) $c_{v,dep}$ is computed by differentiating the equation of departure internal energy (Equation 7.105 (p. 539)) with respect to T, and the partial derivatives of the specific volume are computed by differentiating Equation 7.89 (p. 537) appropriately.

The entropy S is computed by

$$S = S_{ideal,0} - \frac{S_{dep}}{MW} \tag{7.108}$$

$$S_{dep} = -Rln\left(\frac{V-b+c}{V_0}\right) - \frac{\frac{da}{dT}}{\Delta} ln\left[\frac{2V+\delta+\Delta}{2V+\delta-\Delta}\right] \tag{7.109}$$

where

$S_{ideal,0}$ = ideal gas entropy at temperature T and reference pressure (J/kg/K)

V_0 = ideal gas specific molar volume at temperature T and the reference pressure (m^3/kmol)

P = pressure (Pa)

T = temperature (K)

V = specific molar volume (m^3/kmol)

MW = mean molecular weight (kg/kmol)

Δ is given by Equation 7.104 (p. 539) and α, δ, and ε are computed using Equation 7.90 (p. 537) – Equation 7.100 (p. 538), depending on the equation of state model. Note that the pressure term in Equation 7.108 (p. 540) cancels out as both $S_{ideal,0}$ and V_0 are evaluated at the reference pressure.

7.16.3.4. Critical Constants for Pure Components

Equations describing real-gas properties require the knowledge of the critical constants for pure components and mixtures. These comprise the critical temperature (T_c), critical pressure (P_c), critical specific volume (V_c), and the acentric factor (ω).

Several critical constants for fluid materials in the ANSYS Fluent property database `propdb.scm` have been compiled from a variety of sources available in the open literature [87] (p. 2891), [78] (p. 2891), [79] (p. 2891), [107] (p. 2892), [108] (p. 2892), [113] (p. 2893), [7] (p. 2887).

For those fluid materials, for which the critical properties have not been found in the open literature, these have been estimated using the commercially available software CRANIUM by Molecular Knowledge Systems Inc. [1] (p. 2887) : `http://www.molknow.com/Cranium/cranium.htm`

Critical property values for many hydrocarbon and nitrogenous radical species have been obtained from Tang and Brezinsky [118] (p. 2893). Where the critical properties for the radicals were not available in the literature, these were estimated using a modification of the Joback method [87] (p. 2891). This assumes that the radical site constitutes a distinct group with zero group contribution and utilizes the group contribution values for stable species.

The critical properties of coal volatiles have been estimated assuming that the volatiles can be approximated by a mixture of CO, CO_2, H_2, CH_4, and C_2H_6 in such a way, that the atom composition and the net calorific value of the volatiles is similar to that of the assumed mixture. The critical properties of the lignite and biomass volatiles have been assumed equal to those of formaldehyde. The critical properties of diesel and jet-a fuels have been set equal to those of decane. The critical properties of kerosene have been set equal to those of dodecane.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.16.3.5. Calculations for Mixtures

For the computation of properties in real-gas mixtures, ANSYS Fluent follows the so-called pseudocritical method. According to this method, the behavior and properties of a real gas mixture will be the same as that of a pure component, to which appropriate critical constants are assigned. These mixture critical constants are functions of the mixture composition and pure component critical properties, and are sometimes called pseudocritical constants, because their values are generally expected to be different from the true mixture critical constants that may be determined experimentally. However for computational purposes they are the appropriate critical constant values for the mixture. According to the pseudocritical method, ANSYS Fluent applies Equation 7.89 (p. 537)– Equation 7.109 (p. 540) also for mixtures, where the critical temperature T_c, critical pressure P_c, critical specific volume V_c, and acentric factor ω, are replaced by the corresponding mixture critical constants, critical temperature T_{cm}, critical pressure P_{cm}, critical specific volume V_{cm}, and acentric factor ω_m.

The following options are available in ANSYS Fluent for the calculation of the mixture pseudocritical constants:

- The simplest rule for computing the pseudocritical constants C_{cm} for a real gas mixture is the mole fraction average [87] (p. 2891). This method is recommended for mixtures where the pure component critical properties for all components are not very different:

$$C_{cm} = \sum_{i=1}^{k} x_i C_{ci} \tag{7.110}$$

  where

  C_{cm} = mixture pseudocritical constant (temperature, pressure, specific volume or acentric factor)

  C_{ci} = critical constant of component i (temperature, pressure, specific volume or acentric factor)

  x_i = mole fraction of component i

  k = number of components in mixture

- An alternative approach is based on the one-fluid van der Waals mixing rules as expressed in [92] (p. 2892). According to this approach, in order to apply the equation of state models to mixtures, the coefficients a and b in Equation 7.89 (p. 537) are replaced by composition-dependent expressions as follows:

$$(\alpha_m)^{0.5} = \sum_i x_i (\alpha_i)^{0.5} \tag{7.111}$$

$$b_m = \sum_i x_i b_i \tag{7.112}$$

  where

  x_i = mole fraction of species i

  With the appropriate expressions from Equation 7.90 (p. 537)– Equation 7.100 (p. 538) for each equation of state, and assuming a mixture acentric factor ω_m for the evaluation of parameter n in Equation 7.94 (p. 538), Equation 7.97 (p. 538), and Equation 7.99 (p. 538), the mixing rules Equation 7.111 (p. 541) and Equation 7.112 (p. 541) can be rearranged to yield direct expressions of the mixture critical properties as functions of the mole fractions and the component critical properties [8] (p. 2887).

  The resulting expressions for the mixture critical specific temperature T_{cm} are as follows:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– Soave-Redlich-Kwong and Peng-Robinson models

$$T_{cm}=\frac{\left[\sum_i\left(x_i\frac{T_{ci}}{P_{ci}^{0.5}}\right)\right]^2}{\sum_i\left(\frac{x_iT_{ci}}{P_{ci}}\right)} \tag{7.113}$$

– Redlich-Kwong model and Aungier-Redlich-Kwong model:

$$T_{cm}^{1+n}=\frac{\left[\sum_i\left(x_i\sqrt{\frac{T_{ci}^{(2+n)}}{P_{ci}}}\right)\right]^2}{\sum_i\left(\frac{x_iT_{ci}}{P_{ci}}\right)} \tag{7.114}$$

where for the Aungier-Redlich-Kwong model n is obtained from Equation 7.99 (p. 538) as function of the mixture acentric factor ω_m. For the Redlich-Kwong model $n=0.5$.

The mixture critical specific pressure P_{cm} is computed as:

$$P_{cm}=\frac{T_{cm}}{\sum_i\left(\frac{x_iT_{ci}}{P_{ci}}\right)} \tag{7.115}$$

The mixture critical specific volume V_{cm} is computed as:

$$V_{cm}=\sum_I\frac{x_iP_{ci}V_{ci}}{T_{ci}}\left(\frac{T_{cm}}{P_{cm}}\right) \tag{7.116}$$

The notation for Equation 7.113 (p. 542) to Equation 7.116 (p. 542) is

T_{cm} = mixture pseudocritical temperature (K)

P_{cm} = mixture pseudocritical pressure (Pa)

V_{cm} = mixture pseudocritical molar volume (m^3/kmol)

T_{ci} = critical temperature for component i (K)

P_{ci} = critical pressure for component i (Pa)

V_{ci} = critical molar volume for component i (m^3/kmol)

x_i = mole fraction for component i

7.16.3.5.1. Using the Cubic Equation of State Real Gas Models

For single or multicomponent flows, you will enable the cubic equation of state real gas models by selecting **real-gas-soave-redlich-kwong**, **real-gas-peng-robinson**, **real-gas-aungier-redlich-kwong**, or **real-gas-redlich-kwong** from the **Density** drop-down list in the **Create/Edit Materials** dialog box.

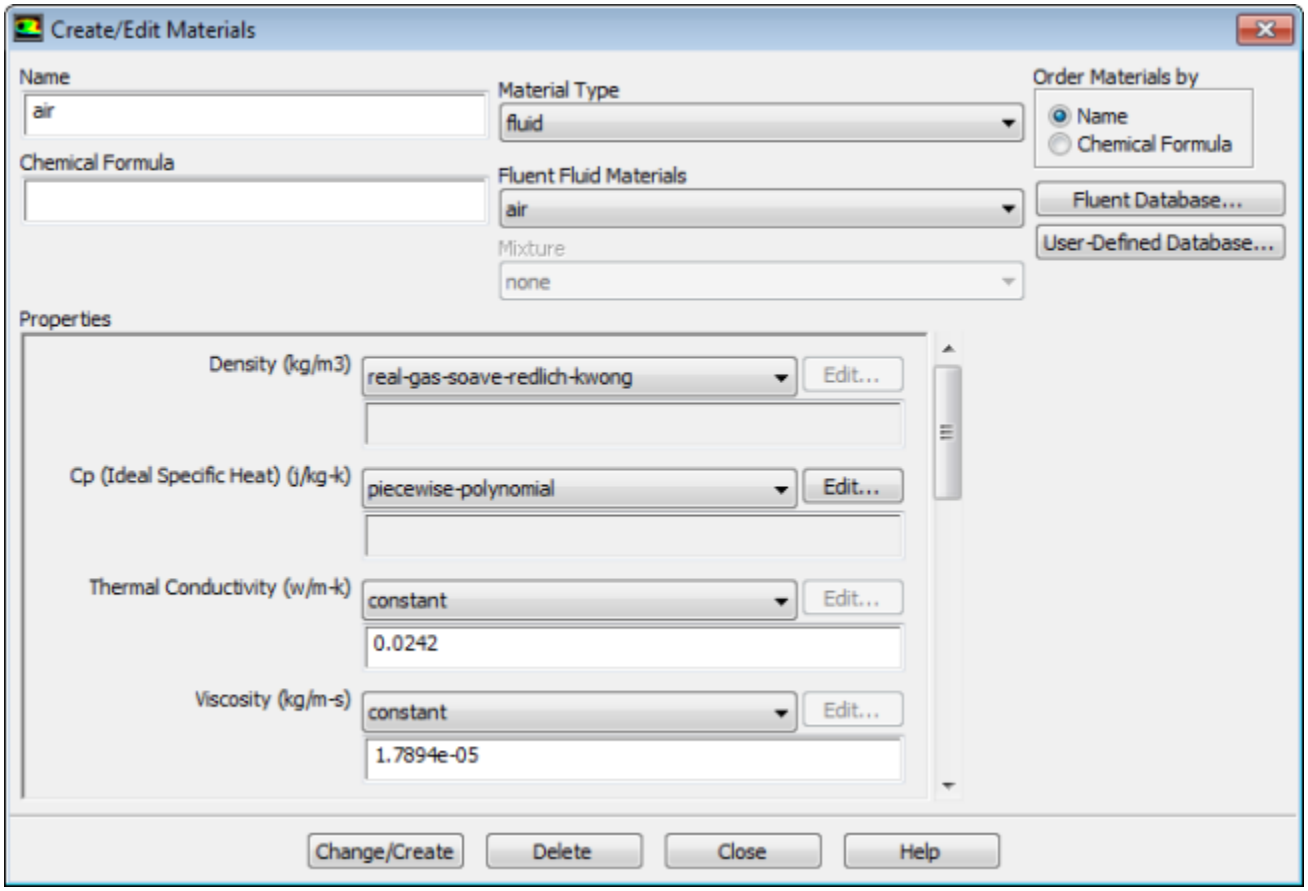
Setup → **Materials** → **Create/Edit...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The required inputs for the cubic equation of state real gas models for single component flow and mixtures are described below.

Single Component Flow

Figure 7.38: The Cubic Equation of State Model for a Real-Gas Fluid



When any of the cubic equation of state models is enabled, enter the following material properties in the dialog box:

- ideal specific heat
- molecular weight
- standard state entropy
- reference temperature
- critical temperature
- critical pressure
- critical specific volume

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

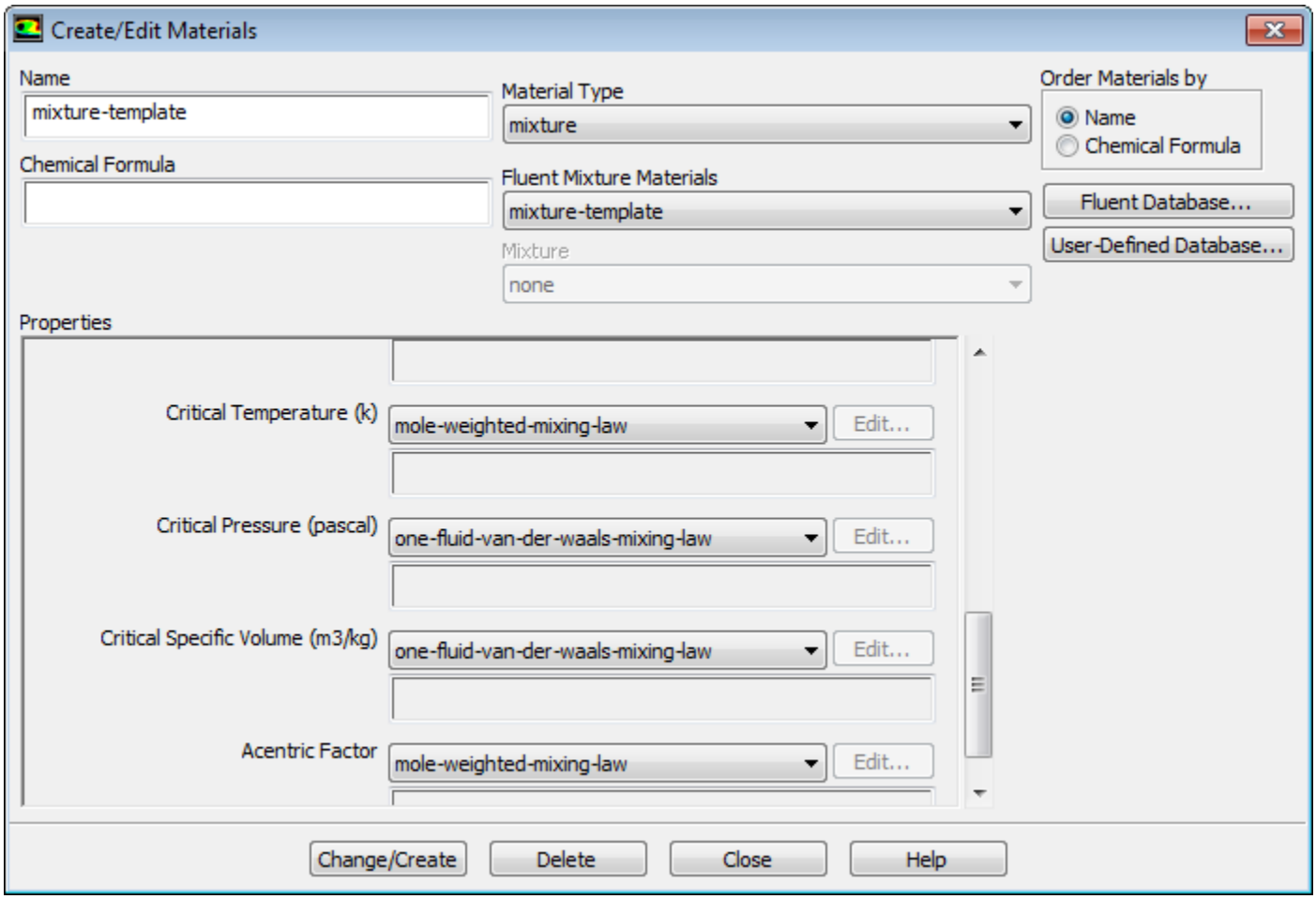
- acentric factor

Important

Your inputs for the specific heat in the **Materials** dialog box will now be used to compute the ideal property functions H_{ideal}, $c_{p,ideal}$, and $S_{ideal,0}$ in Equation 7.102 (p. 538), Equation 7.106 (p. 539), and Equation 7.108 (p. 540), respectively. In ANSYS Fluent the departure properties will be computed and added to the ideal part, to yield the real gas specific heat, enthalpy, and entropy.

Mixtures

Figure 7.39: The Cubic Equation of State Model for a Real-Gas Mixture



When one of the cubic equation of state models is selected from the **Density** drop-down list, specify the following material properties for the mixture material in the dialog box:

- ideal specific heat
- critical temperature
- critical pressure
- critical specific volume

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- acentric factor

You also need to enter the following material properties for each of the mixture components in the dialog box:

- molecular weight
- standard state entropy
- reference temperature

When you are modeling a real-gas mixture, the following methods are available for the mixture critical constants:

- **constant**: defines a constant critical temperature, critical pressure, critical specific volume, or acentric factor for the mixture material.
- **mole-weighted-mixing-law**: applies Equation 7.110 (p. 541) for critical temperature, critical pressure, critical specific volume, or acentric factor for the mixture material.
- **one-fluid-van-der-waals-mixing-law** applies Equation 7.113 (p. 542) and Equation 7.114 (p. 542) for the mixture critical temperature (depending on the real-gas model), Equation 7.115 (p. 542) for the mixture critical pressure, and Equation 7.116 (p. 542) for the mixture molar volume.

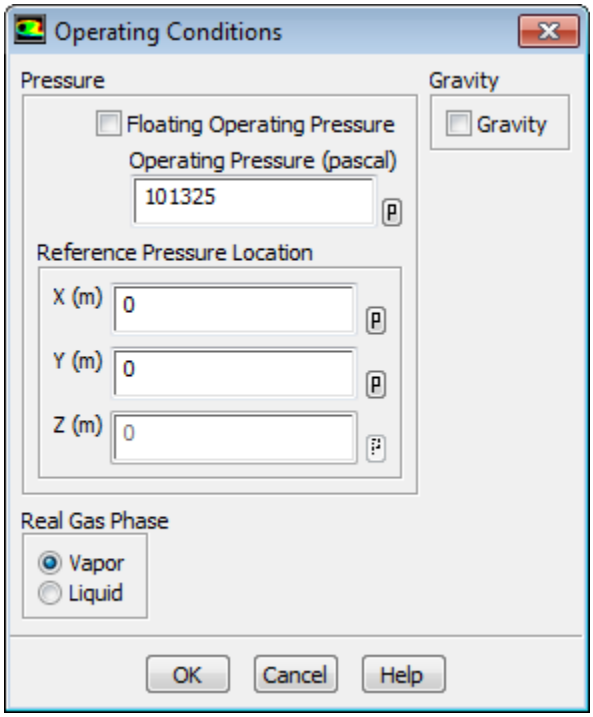
Important

- Ensure to click the **Change/Create** button so that all the above mentioned properties are uploaded and are visible in the interface.
- If you have selected **mixing-law** for the mixture ideal specific heat you will also need to enter the ideal specific heat values for the individual mixture components. If you have not selected **constant** as the option for any of the critical properties, you must enter the corresponding pure component critical properties for the mixture components.

If the operating conditions in your model are in the subcritical regime, select **Vapor** or **Liquid** as the **Real Gas Phase** in the **Operating Conditions** dialog box (Figure 7.40: The Operating Conditions for a Real Gas Phase (p. 546)). Alternatively, you can use the `define/operating-conditions/set-phase` text command. Note that the default is **Vapor** and therefore **Vapor** is assumed if no changes are made to the **Real Gas Phase** settings. If the operating conditions in your model are entirely in the supercritical regime, this setting will have no effect.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.40: The Operating Conditions for a Real Gas Phase



In addition, you may specify the phase for a specific fluid zone by typing the following text command at the ANSYS Fluent console prompt:

```
> define boundary-conditions modify-zones change-zone-phase
Select a name/id from fluid zones list [(liquid-zone fluid-1)] liquid-zone
Set zone real-gas phase:
  -1:use global setting
  0:liquid
  1:vapor
[-1]  0
```

7.16.3.5.2. Solution Strategies and Considerations for Cubic Equations of State Real Gas Models

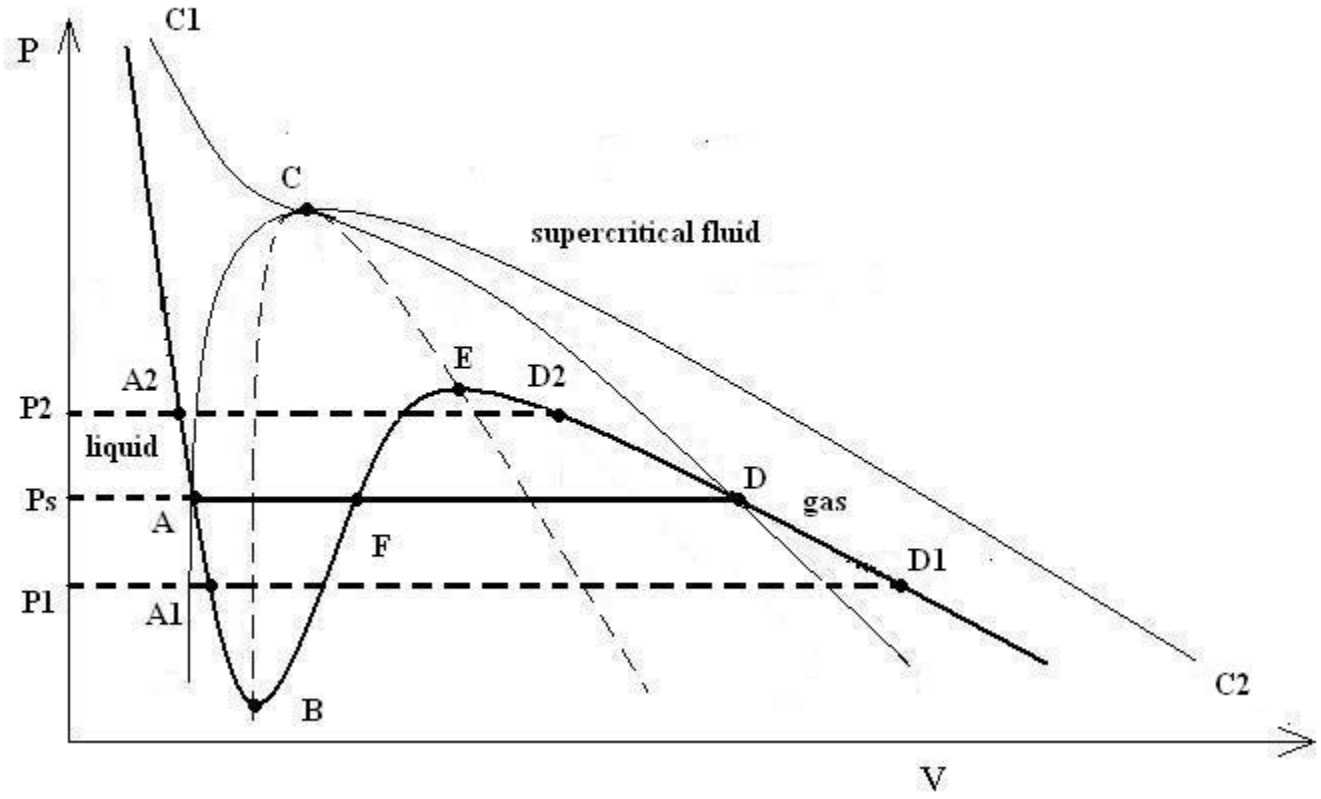
The flow modeling of real-gas flow is more complex and challenging than simple ideal-gas flow. Therefore, the solution might converge at a slower rate with real-gas flow than when running ideal-gas flow. It is recommended that you first attempt to converge your solution using first-order discretization then switch to second-order discretizations and re-iterate to convergence.

Special considerations apply if the operating regime in your model is fully or partly subcritical, where the physical state may be vapor, liquid, or a vapor/liquid mixture. In the cubic EOS real-gas model, the phase state is determined by the selection of the cubic root to calculate the molar-volume. This is illustrated in Figure 7.41: The PV Diagram for the Cubic Equation of State Real Gas Model (p. 547), which shows a pressure versus molar volume (PV) diagram, with a subcritical isotherm ABFED at temperature T calculated from a cubic EOS. Curve C1CC2 represents the critical isotherm and curve ACD the saturation dome. Points A, F, and D represent the three roots of the EOS at the saturation pressure Ps, where A corresponds to the molar volume of saturated liquid, D to the molar volume of saturated vapor and point F does not have a physical significance. Points A1 and D1 correspond to the liquid and vapor molar volumes at pressure P1, which is lower than the saturation pressure. The liquid and vapor molar volumes at pressure P2, which is higher than the saturation pressure, are marked as A2 and D2 respectively. Points B and E, where the partial derivatives of pressure with respect to volume $\left(\partial P/\partial V\right)_T$ are 0,

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

are called spinodal points. The loci of these points for all subcritical temperatures, the spinodal curve, is shown with the dashed curve BCE and sets the boundary beyond which the equation of state is no longer valid, because the local derivative of pressure with respect to volume becomes positive. State points inside the dome, up to the spinodal curve, are called "metastable" because normally they only exist temporarily in small local regions until phase change occurs.

Figure 7.41: The PV Diagram for the Cubic Equation of State Real Gas Model



It is important to realize that the current implementation of the cubic equations of state real gas model does not determine the saturation conditions and does not model the two-phase flow where liquid and vapor coexist. In the subcritical regime and for conditions where the cubic EOS has three roots, the phase state selection is controlled by your input of **Vapor** or **Liquid** for **Real Gas Phase** in the **Operating Conditions** dialog box (Figure 7.40: The Operating Conditions for a Real Gas Phase (p. 546)). The default setting is **Vapor**, which corresponds to the state points to the right of the vapor spinodal. With reference to Figure 7.41: The PV Diagram for the Cubic Equation of State Real Gas Model (p. 547), for the operating point at temperature T and pressure P1, if the **Real Gas Phase** is set to **Vapor**, the molar volume at point D1, which corresponds to superheated vapor, will be selected in the calculation. On the other hand, if you have selected **Liquid**, the molar volume of point A1 will be taken, and the corresponding state will be metastable superheated liquid. Similarly for pressure P2, which is higher than the saturation pressure Ps, if you select **Liquid** for **Real Gas Phase**, the liquid molar volume at A2 will be computed, and if you select **Vapor** the molar volume will correspond to the subcooled vapor state point D2.

In case the calculations fall inside the saturation dome, the solver will not limit the solution, but if conditions are predicted that extend beyond the vapor spinodal curve a warning will be issued:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
temperature is below the spinodal point in 12 cells on zone 3.DE27233
```

Cubic equations of state real gas models are not available for two-phase flows but can be applied to conditions of supercritical pressure $P>P_c$ and subcritical temperature $T<T_c$, where the fluid is in the liquid state, and the supercritical liquid co-exists with gas and supercritical fluid in the same simulation. In those cases, phase change may take place, without any of the flow conditions falling inside the saturation dome.

For multicomponent simulations, when **Diffusion Energy Source** is enabled in the **Species Model** dialog box, the species energy diffusion is by default suppressed in the supercritical liquid regime and across the gas-liquid boundary. The diffusion energy source can be included in the liquid regime using the `define/models/species/liquid-energy-diffusion?` text command.

7.16.3.5.3. Using the Cubic Equation of State Models with the Lagrangian Dispersed Phase Models

If your simulation contains a liquid stream, the appropriate modeling approach in the various operating condition regimes is as follows:

1. For $T>T_c$ a liquid phase does not exist.

2. In the region $P>P_c$, you can define the flow streams directly in the boundary conditions by setting the appropriate pressure and temperature and the properties will be computed directly by the cubic EOS.

3. For $P<P_c$ or $P>P_c$, a stream can exist in liquid phase when its temperature $T<T_c$. If you would like to model phase change in this regime, the following recommendations apply:

   - The DPM droplet and multicomponent models are adequate and recommended for the conditions away from the critical point. This regime can be defined as $T<0.9T_c$, where the liquid physical properties can be assumed independent of pressure.

   - The region $T_c>T>0.9T_c$ is characterized by near-critical-point phenomena, such as strong liquid density and specific heat dependence on both temperature and pressure. The DPM model can also be used in this regime, but you should be cautious, as it will not take into consideration the near-critical-point behavior. In addition, the applicability of the evaporation and boiling rate equations is questionable in this regime.

   - If the pressure in your model is above the critical point, it is recommended to use the **compressible-liquid** method for **Density** of the droplet material.

The temperature $T_{\lim}=0.9T_c$ gives an indication of the maximum droplet temperature for applicability of the DPM models for each droplet material. These temperature limits are listed in Table 7.2: Temperature Limits for Droplet Materials in ANSYS Fluent Database prodb.scm (p. 548) for many of the droplet materials in ANSYS Fluent's `propdb.scm` materials database. For supercritical pressures ($P>P_c$), the DPM models can still be used provided the droplet temperatures remain below the limit $T_{\lim}$.

Table 7.2: Temperature Limits for Droplet Materials in ANSYS Fluent Database prodb.scm

Material	Tc (K)	Pc (MPa)	Normal BP (K)	Tlim (K)
Argon	150.86	4.89	87.30	135.77
Benzene	562.00	4.89	353.00	505.80
Helium	5.30	0.23	4.20	4.77

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Hydrogen	32.98	1.30	20.40	29.68
Methyl-alcohol	512.00	8.10	338.00	460.80
Heptanes	540.00	2.74	371.00	486.80
Hexane	507.00	3.02	342.00	456.30
Octane	569.00	2.49	399.00	512.10
Pentane	470.00	3.37	309.00	423.00
Nitrogen	126.00	3.40	77.40	113.40
Oxygen	154.00	5.04	90.20	138.60
Toluene	592.00	4.11	384.00	532.80
Water	647.00	22.00	373.00	582.30

For high pressure simulations the boiling point will be different from the normal boiling point, and for varying pressure applications the boiling point will vary with the droplet location in the domain.

When a cubic equation of state real gas model is enabled in a simulation that includes evaporating droplet particles, the boiling point is calculated from the vapor pressure data directly, as the temperature where the saturation vapor pressure equals the domain pressure. In addition, the latent heat of the evaporating or boiling droplet will vary with the droplet temperature.

The latent heat at temperature T_p is given by

$$H_{lat_T} = -\int_{T_p}^{T_{bp}} c_{p,g} dT + H_{lat} + \int_{T_p}^{T_{bp}} c_{p,p} dT \tag{7.117}$$

where T_{bp} is the normal boiling point and H_{lat} is the latent heat at the normal boiling point.

In the **Create/Edit Materials** dialog box you must enter the **Normal Boiling Point (NBP)** and the **Latent Heat at NBP** for the **droplet-particle Material Type**. These inputs will be used for calculating the **Latent Heat** at the reference temperature (see Equation 16.437) and the latent heat in the droplet energy balance during vaporization and boiling according to Equation 7.117 (p. 549).

Important

The constant property option is disabled for the **Saturation Vapor Pressure** property when a real gas model is used in the simulation. Also, ensure to enter the appropriate droplet saturation vapor pressure data to cover the complete pressure/temperature range in your model.

Finally when a cubic equation of state real-gas model is enabled in a simulation with droplet models, the condition for switching from the vaporization to the boiling law will be

$$P_{sat} > P \tag{7.118}$$

where P_{sat} is the saturation vapor pressure and P is the domain pressure. If $P_{sat} < P$ while in the boiling law, the model will switch back to the vaporization law. Under supercritical pressure conditions ($P > P_c$), the vaporization models are applicable at droplet temperatures below the critical point, and the switching condition from vaporization to boiling $P_{sat} > P_c$ is never met because the vapor pressure curve is defined only up to the critical point.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.16.3.5.4. Postprocessing the Cubic Equations of State Real Gas Model

All postprocessing functions properly report and display the current thermodynamic and transport properties of the selected real gas model. The thermodynamic and transport properties controlled by the cubic equations of state real gas model include the following:

- density
- enthalpy
- entropy
- sound speed
- specific heat
- any quantities that are derived from the properties listed above (for example, total quantities, ratio of specific heats)

In addition to the properties listed above, you can also report

- compressibility factor
- reduced temperature
- reduced pressure
- spinodal temperature
- subcritical condition

If you are modeling a real-gas mixture you can report the composition-dependent mixture critical properties

- critical temperature
- critical pressure
- critical specific volume
- acentric factor

7.16.4. The NIST Real Gas Models

The NIST real gas models use the National Institute of Standards and Technology (NIST) Thermodynamic and Transport Properties of Refrigerants and Refrigerant Mixtures Database Version 9.1 (REFPROP v9.1) to evaluate thermodynamic and transport properties of approximately 125 fluids or a mixture of these fluids.

The REFPROP v9.1 database is a shared library that is dynamically loaded into the solver when you enable one of the NIST real gas models in an ANSYS Fluent session. Once the NIST real gas model is activated, control of relevant property evaluations is relinquished to the REFPROP database. All postprocessing functions will properly report and display the current thermodynamic and transport properties of the real gas.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.16.4.1. Limitations of the NIST Real Gas Models

The following limitations exist for the NIST real gas model:

- When you are using the NIST real gas models, the NIST materials defined in your simulation will appear in the Create/Edit Materials Dialog Box (p. 2302) either with the name **real-gas-*_name*** , where ***_name*** is the NIST material selected for the single species model, or with the name **real-gas-mixture** for the multiple species model. The inputs for the properties calculated by the NIST functions are disabled. You can use the Create/Edit Materials Dialog Box (p. 2302) to define or modify:
  - Mass diffusivity property in the **real-gas-mixture** material if you are using the multiple-species NIST real gas model (note that the **kinetic-theory** option is not available)
  - Radiation properties if you are modeling radiation
  - Properties of materials other than the NIST **real-gas-mixture** or **real-gas-fluid** materials
- The NIST real gas model assumes that the fluid you will be using in your ANSYS Fluent computation is superheated vapor, supercritical fluid, or liquid. Note that subcritical flow conditions, where vapor coexists with liquid in two-phase flow, are not supported. In addition, all fluid zones must contain the real gas; you cannot include a real gas and another fluid in the same problem.
- Pressure-inlet, mass flow-inlet, and pressure-outlet are the only inflow and outflow boundaries available for use with the real gas models.
- Non-reflecting boundary conditions are not compatible with the real gas models when using the density-based solver. If your model requires NRBC and a real gas model, you must use the pressure-based solver.
- The mixture flow does not permit chemical reactions with the NIST real gas model.
- The real gas models cannot be used with any of the multiphase models. The model is compatible with the Lagrangian Dispersed Phase Models only for the massless and inert particle types.
- You cannot modify material properties in the REFPROP database libraries, or add custom materials to the NIST real gas model.
- The **Diffusion Energy Source** $\nabla \cdot \left(\sum_j h_j \vec{J}_j \right)$ in the energy equation (see The Energy Equation in the Theory Guide) is not included with the nist-multispecies-real-gas-models and the pressure-based solver.

7.16.4.2. The REFPROP v9.1 Database

The NIST real gas model supports 125 fluids from the REFPROP database. These include pure fluids (with the file extention `.fld`) and pseudo-fluids (with the file extention `.ppf`). The fluids that are supported by REFPROP v9.1 and used in the NIST real gas model are listed in Table 7.3: Fluids Supported by REFPROP v9.1 (p. 552) (the corresponding property file name appears in parentheses, where it does not coincide with the fluid name).

The REFPROP v9.1 database employs accurate pure-fluid equations of state that are available from NIST. These equations are based on three models:

- modified Benedict-Webb-Rubin (MBWR) equation of state
- Helmholtz-energy equation of state

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- extended corresponding states (ECS)

For a fluid that consists of a multispecies-mixture the thermodynamic properties are computed by employing mixing-rules applied to the Helmholtz energy of the mixture components.

Important

The database does not include transport property models for the species marked with * in Table 7.3: Fluids Supported by REFPROP v9.1 (p. 552). As a result the NIST real gas model with those species can only be used for modeling inviscid flow.

Table 7.3: Fluids Supported by REFPROP v9.1

1-butene	nitrogen
propanone (acetone.fld)	nonane
air	Dodecafluoro-2-methylpentan-3-one (novec649.fld)
ammonia	octane
argon	orthohydrogen (orthohyd.fld)*
benzene	oxygen
n-butane (butane.fld)	1,2-dimethylbenzene (oxylene.fld)*
dodecane (c12.fld)	parahydrogen (parahyd.fld)
methylcyclohexane (c1cc6.fld)	pentane
cis-2-butene (c2butene.fld)	propane
n-propylcyclohexane (c3cc6.fld)	propene (propylen.fld)
decafluorobutane (c4f10.fld)	propyne
dodecafluoro-pentane (c5f12.fld)	1,4-dimethylbenzene (pxylene.fld)*
trifluoro-iodo-methane (cf3i.fld)	1,1,2-trichloro-1,2,2-trifluoro-ethane (r113.fld)
carbondioxide	1,2-dichloro-1,1,2,2-tetrafluoro-ethane (r114.fld)
carbonmonoxide	chloro-pentafluoro-ethane (r115.fld)
carbonoxidesulfide (cos.fld)*	hexafluoro-ethane (r116.fld)
cyclohexane (cyclohex.fld)	trichloro-fluoro-methane (r11.fld)
cyclopentane (cyclopen.fld)	hexafluoro-propene (r1216.fld)
cyclopropane (cyclopro.fld)	1-chloro-3,3,3-trifluoroprop-1-ene (r1233zd.fld)
deuterium (d2.fld)	2,3,3,3-tetrafluoroprop-1-ene (r1234yf.fld)
deuteriumoxide (d2o.fld)	trans-1,3,3,3-tetrafluoro-propene (r1234ze.fld)
octamethyl-cyclotetra-siloxane (d4.fld)	2,2-dichloro-1,1,1-trifluoro-ethane (r123.fld)
decamethyl-cyclopentasiloxane(d5.fld)	1-chloro-1,2,2,2-tetrafluoro-ethane (r124.fld)
dodecamethyl-cyclohexasiloxane (d6.fld)	pentafluoro-ethane (r125.fld)
decane	dichloro-difluoro-methane (r12.fld)
diethylether (dee.fld)*	1,1,1,2-tetrafluoro-ethane (r134a.fld)
dimethylestercarbonicacid (dmc.fld)	chlorotrifluoro-methane (r13.fld)
methoxymethane (dme.fld)	1,1-dichloro-1-fluoroethane (r141b.fld)
phenylethane (ebenzene.fld)*	1-chloro-1,1-difluoro-ethane (r142b.fld)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ethane	1,1,1-trifluoroethane (r143a.fld)
ethyl alcohol (ethanol.fld)	tetrafluoromethane (r14.fld)
ethene (ethylene.fld)	1,1-difluoroethane (r152a.fld)
fluorine	fluoroethane (r161.fld)
hydrogen sulfide (h2s.fld)	octa-fluoro-propane (r218.fld)
hydrogen chloride (hcl.fld)*	dichloro-fluoro-methane (r21.fld)
helium	1,1,1,2,3,3,3-hepta-fluoro-propane (r227ea.fld)
heptane	chloro-difluoro-methane (r22.fld)
hexane	1,1,1,2,3,3-hexafluoro-propane (r236ea.fld)
hydrogen (normal)	1,1,1,3,3,3-hexafluoro-propane (r236fa.fld)
2-methyl-1-propene (ibutene.fld)	trifluoro-methane (r23.fld)
2-methylpentane (ihexane.fld)	1,1,2,2,3-pentafluoro-propane (r245ca.fld)
2,2,4-trimethylpentane (ioctane.fld)	1,1,1,3,3-pentafluoro-propane (r245fa.fld)
2-methylbutane (ipentane.fld)	difluoro-methane (r32.fld)
2-methylpropane (isobutan.fld)	1,1,1,3,3-pentafluoro-butane (r365mfc.fld)
krypton	44% R125/4% R134a/52% R143a (r404a.ppf)*
decamethyl-tetrasiloxane (md2m.fld)	23% R32/25% R125/52% R134a (r407c.ppf)*
dodecamethyl-pentasiloxane (md3m.fld)	methyl chloride (r40.fld)*
tetradecamethyl-hexasiloxane (md4m.fld)	50% R32/50% R125 (r410a.ppf)*
octamethyl-trisiloxane (mdm.fld)	fluoro-methane (r41.fld)
methane	50% R125/50% R143a (r507a.ppf)*
methanol	octafluoro-cyclobutane (rc318.fld)
methyl (Z,Z)-9,12-octadecadienoate (mlinolea.fld)	methyl trifluoro-methyl ether (re143a.fld)*
methyl (Z,Z,Z)-9,12,15-octadecatrienoate (mlinolen.fld)	methyl-pentafluoro-ethyl-ether (re245cb2.fld)
hexamethyl-disiloxane (mm.fld)	2,2,2-trifluoroethyl-difluoromethyl-ether (re245fa2.fld)
methyl cis-9-octadecenoate (moleate.fld)	methyl-heptafluoro-propyl-ether (re347mcc.fld)
methyl hexadecanoate (mpalmita.fld)	sulfur hexafluoride (sf6.fld)
methyl octadecanoate (mstearat.fld)	sulfur dioxide (so2.fld)
1,3-dimethylbenzene (mxylene.fld)	trans-2-butene (t2butene.fld)
dinitrogen monoxide (n2o.fld)	methylbenzene (toluene.fld)
neon	water
2,2-dimethylpropane (neopentn.fld)	xenon
nitrogen trifluoride (nf3.fld)*	

7.16.4.3. Using the NIST Real Gas Models

When you enable one of the NIST real gas models (pure fluid or multiple-species mixture) and select a valid material, ANSYS Fluent's functionality remains the same as when you model fluid flow and heat transfer using an ideal gas, with the exception of the Create/Edit Materials Dialog Box (p. 2302).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.16.4.3.1. Activating the NIST Real Gas Model

Activating one of the NIST real gas models is a two-step process. First you enable either the single- or multi-species NIST real gas model, and then you select the fluid material from the REFPROP database.

1. Enable the appropriate NIST real gas model by typing the following text command at the ANSYS Fluent console prompt:

   - Pure fluid real gas model:

     ```
     > define/user-defined/real-gas-models/nist-real-gas-model
     use NIST real gas? [no] yes
     ```

   - Multi-species real gas model:

     ```
     > define/user-defined/real-gas-models/nist-multispecies-real-gas-model
     use multispecies NIST real gas? [no] yes
     ```

   The list of available pure fluid materials you can select from will be displayed.

2. Select material from the REFPROP database list:

   - Pure fluid real gas model:

     Enter the name of a single pure fluid that you want to investigate as follows:

     ```
     select real-gas data file [""]  methane.fld
     ```

   - Multi-species real gas model:

     Enter the number of desired species in the mixture:

     ```
     Number of species []  3
     ```

     followed by the name of each fluid in quotation marks selected from the list printed in the console:

     ```
     select real-gas data file [""] nitrogen.fld
     select real-gas data file [""] co2.fld
     select real-gas data file [""] methane.fld
     ```

     a. Upon selection of a valid pure or multi-species fluids, ANSYS Fluent will load the relevant material data from a library of pure fluids supported by the REFPROP database, and report that it is opening the shared library (`librealgas.so`) where the compiled REFPROP database source code is located. A list of properties for the specified material, including its applicable pressure and temperature ranges, will also be reported in the console.

     Important

     – For mixture flows, not all combinations of species mixtures are allowed. This could be due to lack of data for one or more binary pairs. In such situations an error message generated by NIST will be returned and displayed on the ANSYS Fluent console, and no real gas material is allowed to be created. In some combinations the mixing data will be estimated, a warning message will be displayed on the ANSYS Fluent console and the mixture material allowed to be created.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– Transport property calculations are not supported for mixtures that include water with molar concentration over 5%. If this limit is exceeded in your calculation, ANSYS Fluent will issue a warning message. Transport equations are not available for the ammonia/water mixture and for mixtures with alcohols. As a result, the NIST real gas model with those mixtures can only be used for modeling inviscid flow.

3. If you want to use NIST lookup tables in your simulation, you can create them as described in the following sections:

   • Creating Full NIST Look-up Tables (p. 555) (single and multi-species materials with thermal properties as well as saturation data for a fixed multi-species composition)

   • Creating Binary Mixture Saturation Tables for Binary Mixtures (p. 558) (binary mixtures only)

4. (cases with no NIST tables only) If you would like to model flow in the liquid phase, you can use the `set-phase` TUI command. Note that the default phase is vapor, so if you do not go through this step, vapor is assumed. Also, if the flow conditions do not permit liquid to exist, a vapor calculation will be performed instead.

```
  > define/user-defined/real-gas-models/set-phase
Select vapor phase (else liquid)?[yes]
```

   In addition, you may specify the phase for a specific fluid zone by typing the following text command at the ANSYS Fluent console prompt:

```
> define boundary-conditions modify-zones change-zone-phase
Select a name/id from fluid zones list [(liquid-zone fluid-1)] liquid-zone
Set zone real-gas phase:
  -1:use global setting
  0:liquid
  1:vapor
[-1]  0
```

7.16.4.3.2. Creating Full NIST Look-up Tables

By default, the NIST functions are called directly each time the thermal properties are updated during iteration. Alternatively, you can use the NIST look-up tables in your simulation as described below. Computing the look-up table offers two advantages:

• Reduction in computation time because the NIST functions do not need to be called each time the thermal properties are updated

• The look-up table created is applicable to both liquid and vapor phases

To create a full NIST look-up table, first enable the NIST real gas model as described in Activating the NIST Real Gas Model (p. 554), and then follow these steps:

1. When prompted with `Create Full NIST LookUp Tables for multi components? [no]`, answer **yes**.

2. Specify parameters for the look-up table generation.

   For both single- and multi-species real gas models, you need to specify:

   • Pressure and temperature bounds for the table

You must ensure that the temperature and pressure bounds you specify are strictly within the stated limits of applicability that are displayed when the NIST model is loaded.

- Number of pressure, temperature, and saturation points, which determines the resolution of the property look-up table

  The minimum number of pressure and temperature points is 12 and there is no upper limit. However, the larger the table, the greater the memory and computing time requirements. Use a few test runs to strike the best balance of costs (memory and computing time) and accuracy.

For multi-species, you will also need to specify:

- Upper and lower molar density bounds for bubble and dew point calculations

  Molar density is used to build the saturation table. The default values work well for most compositions.

- The mixture composition on a mass or mole basis

  You must ensure that the sum of all constituent fractions adds up to 1 when specifying mass/molar fractions for the components. The maximum number of species is 20.

- The bubble points and dew points filenames and the directory where you want to store the files.

Examples of inputs for the NIST look-up tables:

- Pure fluid real gas model:

```
Min Pressure Value For NIST Table (pa) [13200]
Max Pressure Value For NIST Table (pa) [20000] 1000000
Min Temperature Value For NIST Table (k) [237.3]
Max Temperature Value For NIST Table (k) [500] 400
Number of Pressure Points For NIST Table [101]
Number of Temperature Points For NIST Table [101]
Number of Saturation Points For Bubble/Dew Curve [300]

 NIST table created! Properties for both liquid and vapor phases included!
 Tcrit = 3.391730e+02 pcrit = 3.617700e+06 Dcrit = 5.735823e+02 Ttrp = 1.725200e+02
```

- Multi-species real gas model:

```
Min Pressure Value For NIST Table (pa) [130000]
Max Pressure Value For NIST Table (pa) [5000000]
Min Temperature Value For NIST Table (k) [220] 180
Max Temperature Value For NIST Table (k) [490] 600
Number of Pressure Points For NIST Table [101]
Number of Temperature Points For NIST Table [101]
Please Select Mass Fractions [Y] or Molar Fractions [N] for Composition of Fluid
 [yes]
Composition of Fluid will be set in Mass Fractions!
co2.fld [0.4]
methane.fld [0.4] 0.6
Create NIST Saturation Curves for the same multi components?
[no] y
Number of Saturation Points For Bubble/Dew Curve [300] 22
Min Molar Density For Bubble and Dew Points Calculations (mol/l) [0.1]
Max Molar Density For Bubble and Dew Points Calculations (mol/l) [25]
Bubble Points File Name ["Bubble_PT.xy"]
Dew Points File Name ["Dew_PT.xy"]
Directory to store Saturation Files [""]
Start NIST Look-Up Table Calculations ...

Property Table... 83.3 percent completed!
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
NIST table for 2-components created! Properties for both liquid and vapor phases included!
End NIST Look-Up Table Calculations
Use the created NIST Lookup Table for thermal property calculations? [no] y
```

Once the table is created, you must confirm that you want to use the look-up table for thermal properties calculations when prompted.

```
Use the created NIST Lookup Table for thermal property calculations? [no] yes

 The created NIST Lookup Table will be used to calculate the thermal properties!
```

If properties are needed at conditions outside the limits you have specified for the table, Fluent will automatically call the original NIST model functions to calculate the thermal properties. If more than 10% of the property evaluations require computation outside the look-up table, Fluent will print a message indicating the percentage of calculations using direct computation. This can be used as an indicator to adjust the temperature and pressure ranges for the table to minimize the computational cost.

For multi-species real gas properties, the following tables will be created:

- fluid properties
- saturation conditions

Remarks and Limitations on Using Multi-Species Full NIST look-up Tables

- The look-up table is not saved with the case file. If you restart Fluent or re-read case and data files into the current Fluent session, the table will be cleared and you must go through the steps above to re-create it before running a simulation.
- For computation of saturation properties for the bubble and dew points for a single- or multiple-species, Fluent provides macros that you can use in UDFs. See NIST Real Gas Saturation Properties in the *Fluent Customization Manual* for more information.
- The tables apply for compressible fluids with material properties that are a function of local pressure and temperature. The material properties covered by the table include fluids in the liquid and vapor phases. The flashing applications between liquid and vapor phases can be conducted using saturation conditions established for the fluid(s) using mass transfer models.
- The property look-up tables and saturation tables are created for fixed fluid compositions. This means the look-up tables are based on the following assumptions:
  - Mixture composition is constant
  - No chemical reactions are involved
  - No mixing of fluids of different compositions, including no mixing between phases
- The look-up table only applies to single-species/multi-species in single-phase flows.
- The temperature and pressure bounds must be specified within the stated limits of applicability that are displayed when the NIST model is loaded.
- When specifying the table size, it is important to pick temperature and pressure ranges that are large enough to satisfy the current flow conditions.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If the temperature and pressure ranges you specify are too narrow, Fluent uses the NIST model to calculate the thermal properties. The time saved by creating the table will be negated if the percentage of calculations performed outside of the table is greater than 10%. This percentage is printed, so you can use it as an indicator for adjusting the temperature and pressure range of the table.
- For multi-species table, the bubble and dew points are saved into files(Fluent plot file format) called `Bubble_PT.xy` and `Dew_PT.xy` in the working directory.
- The look-up table is a structured table and may not be suitable for simulating flow behaviors near the critical point when higher resolution is required.

7.16.4.3.3. Creating Binary Mixture Saturation Tables for Binary Mixtures

To create a saturation data look-up table for binary species mixtures (mixtures of two species only), first enable the NIST real gas model as described in Activating the NIST Real Gas Model (p. 554) and then follow these steps:

1. When prompted with `Create Full NIST LookUp Tables for multi components? [no]`, answer **no**.
2. When prompted with `Create Saturation Table For Binary Mixture Only? [no]`, answer **yes**.

   Important

   Not all binary mixtures can be computed from the Fluent built-in NIST model. The allowed combination of materials is determined by the current database of NIST that can be found in `hmx.bnc` in the ANSYS Fluent installation directory.

3. Specify parameters for the binary saturation data look-up table generation:
   - The number of saturation curves
   - The number of points for each saturation curve

     The default number is 12. The actual number of points for each bubble and dew curve may be different than specified because the NIST model may not yield valid computation results for every density point for a given mass fraction.
   - Upper and lower molar density bounds for bubble and dew point calculations

     The molar density is used to build the saturation table. You should provide reasonable values for the minimum and maximum molar densities corresponding to gas phase at low pressure and high temperature, and liquid phase at high pressure and low temperature. The default values (as shown in the example below) are a good starting point. You may need to adjust these values if there is not enough data in the selected range.
   - The filenames for bubble points and dew points and the existing directory where you want to store the files.

   An example of inputs for the NIST binary look-up table:

```
Number of Mass Fraction Points For Binary Mixture [12]
Number of Saturation Points For Bubble/Dew Curve [12]
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Min Molar Density For Bubble and Dew Points Calculations (mol/l) [0.1]
Max Molar Density For Bubble and Dew Points Calculations (mol/l) [25]
Bubble Points File Name ["Bubble_PT.xy"]
Dew Points File Name ["Dew_PT.xy"]
Directory to store Saturation Files [""] out
```

The lookup table builds multiple sets of the bubble and dew points for the binary mixture with mass fractions for the first component between 0 and 1. The second component mass fraction is equal to the first component mass fraction subtracted from unity. The number of saturation curves is determined by the number of mass fraction points that you specified.

Once the lookup table is created, Fluent saves the bubble and dew points data into files and directory you specified.

NIST Binary Saturation Files

NIST binary saturation files store saturation data lookup table for bubble and dew points for a binary mixture with mass fractions between 0 and 1. They can be used for mass and heat transfer modelling involving phase changes.

The binary saturation file structure is as follows. The file contains several data sets for bubble/dew point curves for a series of mass fractions. Each data set consists of the following three parts:

1. Line 1: The mass fraction of the first species (a single float number between 0 and 1).
2. Line 2: The number of data pairs for saturation pressure and temperature.
3. The specified number of lines that follow line 2: Data pairs of saturation pressure and temperature. The last pressure-temperature pair corresponds to the critical condition.

The example below shows the saturation dew point data for the first two curves, both with 12 pressure-temperature pairs. The first data set is for 0 and the second is for a 0.083333 mass fraction of the first species. The pressure-temperature critical condition is (4.872200e+06, 3.053220e+02) for the first curve and (5.547305e+06, 2.947561e+02) for the second curve.

```
 0.000000
12
1.142108e+00 9.036800e+01
7.314020e+01 1.099093e+02
1.202771e+03 1.294505e+02
8.833626e+03 1.489918e+02
3.890130e+04 1.685331e+02
1.219542e+05 1.880744e+02
3.025174e+05 2.076156e+02
6.354397e+05 2.271569e+02
1.182515e+06 2.466982e+02
2.011733e+06 2.662395e+02
3.201942e+06 2.857807e+02
4.872200e+06 3.053220e+02

 0.083333
12
1.509489e+05 1.891238e+02
1.257388e+06 2.428513e+02
2.252097e+06 2.631615e+02
3.090473e+06 2.752623e+02
3.775539e+06 2.831498e+02
4.318958e+06 2.883822e+02
4.735586e+06 2.917723e+02
5.042345e+06 2.938262e+02
5.258091e+06 2.949048e+02
5.402834e+06 2.952901e+02
5.495209e+06 2.951982e+02
```

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
 5.547305e+06 2.947561e+02

 .....
 .....
```

7.16.4.3.4. Changing the REFPROP Library and Fluid Files

In ANSYS Fluent, you can change the REFPROP library and the fluid files that are accessed by the NIST real-gas model by entering the following text command in the ANSYS Fluent console:

`/define/user-defined/real-gas-models/nist-settings`

When prompted with

```
User-defined refprop library and fluids files?
```

answer **yes**.

The default names for the refprop library and the fluid file paths correspond to the REFPROP v9.1 files provided with your ANSYS Fluent installation. You can revert to the older REFPROP v7.0.1 database by accepting the default refprop library path and entering the following names at the prompts for library name and fluid files path:

```
Select refprop library path ["path\fluent18.1.0\realgas"]
Select refprop library name[“librealgas”] “librealgas7”
Select refprop fluid files path ["path\fluent18.1.0\realgas\lib"] “path\fluent18.1.0\realgas\lib7.0”
```

where *path* is the ANSYS Fluent installation directory (for example, `C:\Program Files\ANSYS Inc\v181\fluent`).

ANSYS Fluent will open REFPROP v7.0 library and report the information in the console.

To switch back to the default REFPROP v9.1 property database, enter the `/define/user-defined/real-gas-models/nist-settings` command again and enter **no** at the prompt. ANSYS Fluent will report the change in the console window.

7.16.4.4. Solution Strategies and Considerations for NIST Real Gas Model Simulation

The flow modeling of NIST real-gas flow is much more complex and challenging than simple ideal-gas flow. Therefore, you should expect the solution to converge at much slower rate with real-gas flow than when running ideal-gas flow. Also due to the complexity of the equations used in property evaluations, converging a solution with the real-gas model is in general done at much lower Courant values when you are using the density-based solver, or at much lower under-relaxation values if you are using the pressure-based solver. It is recommended that you first attempt to converge your solution using first-order discretization, then switch to second-order discretizations and re-iterate to convergence.

The real-gas properties in NIST are defined within a limited/bounded range. It is important that the flow conditions you are prescribing fall within the range of the database. It is possible that you specify flow at a state that is physically valid but otherwise not defined in the database. In this situation the solution will diverge or immediately generate an error message on the ANSYS Fluent console as soon as the state crosses the limit of the database. In some instances, the actual converged state is just within the bounded defined database but only transitory outside the range. In this situation the divergence can be avoided by lowering the Courant value or under-relaxation factors so a less aggressive convergence rate is adapted.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Finally, if you attempt to initialize the flow from an inlet flow conditions and an error message is generated from one of the property routines, then this is an indicator that the flow conditions you have specified is not defined within the range of the database.

7.16.4.4.1. Writing Your Case File

When you save your completed real gas model to a case file, the linkage to the shared library containing real gas properties will be saved to the case file (along with property data for the material you selected in the NIST real gas model). Consequently, whenever you read your case file in a later session, ANSYS Fluent will load and report this information to the console during the read process.

7.16.4.4.2. Postprocessing

All postprocessing functions properly report and display the current thermodynamic and transport properties of the selected real gas model. The thermodynamic and transport properties controlled by the NIST real gas model include the following:

- density
- enthalpy
- entropy
- molecular weight
- molecular viscosity
- sound speed
- specific heat
- thermal conductivity In addition to the properties listed above, you can also report
- compressibility factor
- any quantities that are derived from the properties listed above (for example, total quantities, ratio of specific heats)

7.16.5. The User-Defined Real Gas Model

The user-defined real gas model (UDRGM) has been developed to allow you to write your own custom real gas model to fit your particular modeling needs. It also allows you to simulate a single-species flow, multiple-species mixture flow, multiphase flow, or volumetric reactions.

The following limitations exist for the UDRGM:

7.16.5.1. Limitations of the User-Defined Real Gas Model

- You cannot include more than one user-defined real gas material (fluid or mixture) in the same problem. However, you can use other materials together with real gas in your simulation.
- When you are using the UDRGM, the materials defined in your real gas UDF will appear in the Create/Edit Materials Dialog Box (p. 2302) with the name **real-gas-fluid** or **real-gas-mixture** and all physical and thermodynamic property inputs disabled. Use the Create/Edit Materials Dialog Box (p. 2302) to define or modify:

  - Mass diffusivity property in the real-gas-mixture material if you are modeling multicomponent flow.
  - Chemical reactions in the real-gas-mixture if you are modeling reacting flow.
  - Radiation properties for the real-gas-mixture or the real-gas-fluid materials if you are modeling radiation.
  - Properties of materials other than the user-defined **real-gas-mixture** or **real-gas-fluid** materials.

  Note

  If you are using the UDRGM together with other materials from ANSYS Fluent's property database in dispersed phase or multiphase calculations, take care to use the same reference temperature as in ANSYS Fluent in your real-gas UDF. The reference temperature in ANSYS Fluent is 298.15 K.

- Pressure-inlets, mass flow-inlets, and pressure-outlets are the only inflow and outflow boundaries available for use with the real gas models.
- Non-reflecting boundary conditions are not compatible with the real gas models when using the density-based solver. If your model requires NRBC and a real gas model, you must use the pressure-based solver.
- The UDRGM can be used with the Eulerian multiphase models.

  Note

  Note the following restrictions:

  - Only one phase can be a real gas.

- The UDRGM is compatible with the Lagrangian Dispersed Phase Models. Refer to Using the Cubic Equation of State Models with the Lagrangian Dispersed Phase Models (p. 548) for guidelines and restrictions of this approach.
- The real gas models cannot be used with the non-premixed, partially premixed, and composition PDF transport combustion models. Chemical reactions can however be modeled with the finite rate and eddy dissipation models. Note that the **Dimension Reduction** model is not available with the real gas models.

The UDRGM requires a library of functions written in the C programming language. Moreover, there are certain coding requirements that need to be followed when writing these functions. Sample real gas function libraries are provided to assist you in writing your own UDRGM. When UDRGM functions are compiled, they will be grouped in a shared library that later will be loaded and linked with the ANSYS Fluent executable. The procedure for using the UDRGM is defined as follows:

1. Define the real gas equation of state and all related thermodynamic and transport property equations.
2. Create a C source code file that conforms to the format defined in this section.
3. Start ANSYS Fluent and set up your case file in the usual way.
4. Compile your UDRGM C library and build a shared library file (you can use the available compiled UDF utilities in either the graphical user interface or the text command interface).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Load your newly created UDRGM library via the text command menu:

   If a single-species UDRGM to be used, then the text command menu is:

```
> define/user-defined/real-gas-models/user-defined-real-gas-model

  use user defined real gas? [no] yes
```

   On the other hand, if you are simulating multiple-species UDRGM flow, then the text command menu to use is:

```
> define/user-defined/real-gas-models/user-defined-multispecies-real-gas-model
  use user multispecies defined real gas? [no] yes
```

   Upon activating the UDRGM, the function library will now supply the fluid material properties for your case.

6. You can simulate volumetric reactions with your real gas model using the Species Model Dialog Box (p. 2216), or the text interface (`define/models/species/volumetric-reactions?`).

   You can access the **Species Model** from the tree by going to **Setup**/**Models**/**Species** and double-clicking **Species**.

   In the Species Model Dialog Box (p. 2216)

   - Enable **Species Transport** under **Model**.
   - Enable **Volumetric** under **Reactions**.
   - Select the appropriate **Turbulence-Chemistry Interaction** option.
   - Set up the reaction by clicking the **Edit...** button for the **real-gas-mixture Mixture Material**.

   Important

   Note that the fluid materials and their properties, appearing in the Create/Edit Materials Dialog Box (p. 2302), are the ones defined in your real gas UDF. You cannot modify the materials via this dialog box, however, you can set up the volumetric reaction. If you would like to modify the mixture materials and their properties, this should be done in the real gas UDF. The volumetric reactions for your real gas mixture are defined in the same way as for any ANSYS Fluent mixture. For details, refer to Defining Reactions (p. 977).

   Alternatively, the chemical reactions can be set up using the `define/models/species` and `define/materials` text command.

   Important

   Note that the chemical reactions should be activated after your real gas UDF has been built and loaded. It is also recommended to test and validate your real gas UDF, running the cold flow calculation prior to attempting to solve the reacting flow. Also, make sure that the applicability range of the real gas functions in your UDF fully covers the temperature and pressure range of the reacting flow calculation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Run your calculation.

When using the UDRGM the robustness of the solver and the speed of flow convergence will largely depend on the complexity of the material properties you have defined in your UDF. It is important to understand the operational range of the property functions you are coding so you can simulate the flow within that range.

7.16.5.2. Writing the UDRGM C Function Library

Creating a UDRGM C function library is reasonably straightforward; however, your code must make use of specific function names and macros, which will be described in detail below. The basic library requirements are as follows:

- The code must contain the `udf.h` file inclusion directive at the beginning of the source code. This allows the definitions for `DEFINE` macros and other ANSYS Fluent functions to be accessible during the compilation process.
- The code must include at least one of the UDF's `DEFINE` functions (that is `DEFINE_ON_DEMAND`) to be able to use the compiled UDFs utility (see the sample UDRGM codes provided below).
- Any values that are passed to the solver by the UDRGM or returned by the solver to the UDRGM are assumed to be in SI units.
- You must use the principal set of functions listed below in your UDRGM library. These functions are the mechanism by which your thermodynamic property data are transferred to the ANSYS Fluent solver. Note that `ANYNAME` can be any string of alphanumeric characters, and allows you to provide unique names to your library functions.

Function inputs from the ANSYS Fluent solver consist of one or more of the following variables:

T = Temperature, K

p = Pressure, Pa

ρ = Density, kg/m^3

$Y_i[]$ = Species mass fraction

Important

$Y_i[]$: ANSYS Fluent solver returns a value of `1.0` for $Y_i[]$ in single-species flows. For multiple-species flows, $Y_i[]$ is a vector array containing species mass fraction in an order defined by the user setup function.

The UDRGM function names and argument lists, followed by a short description of the function, are as follows:

`void ANYNAME_error(int err, char *f, char *msg)`
prints error messages.

`void ANYNAME_Setup(Domain *domain, cxboolean vapor_phase, char *filename,int (*messagefunc)(char *format,...), void (*errorfunc)(char *format, ...))`
performs model setup and initialization. Can be used to read data and parameters related to your UDRGM. When writing UDFs for multiple-species, use this function to specify the number of species and the name

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of the species as shown in the multiple-species example. The Boolean variable, `vapor_phase`, passes to your UDF the setting of the text-interface command `define/user-defined/real-gas-models/set-phase`.

`double ANYNAME_density(cxboolean vapor_phase, double T, double P, double yi[])`

returns the value of density as a function of phase, temperature, pressure and species mass-fraction if applicable. The Boolean variable `vapor_phase` passes to your UDF the setting of the text-interface command `define/user-defined/real-gas-models/set-phase`, or, if applicable, the zone phase set by the text-interface command `define/boundary-conditions/modify-zones/change-zone-phase`.

Important

Since this function is called numerous times during each solver iteration, it is important to make this function as numerically efficient as possible.

`double ANYNAME_specific_heat(double T, double Rho, double P, double yi[])`

returns the real gas specific heat at constant pressure as a function of temperature, density, absolute pressure, and species mass-fraction if applicable.

`double ANYNAME_enthalpy(double T, double Rho, double P, double yi[])`

returns the enthalpy as a function of temperature, density, absolute pressure, and species mass-fraction if applicable.

`double ANYNAME_entropy(double T, double Rho, double P, double yi[])`

returns the entropy as a function of temperature, density, absolute pressure, and species mass-fraction if applicable.

`double ANYNAME_mw(double yi[])`

returns the fluid molecular weight.

`double ANYNAME_speed_of_sound(double T, double Rho, double P, double yi[])`

returns the value of speed of sound as a function of temperature, density, absolute pressure, and species mass-fraction if applicable.

`double ANYNAME_viscosity(double T, double Rho, double P, double yi[])`

returns the value of dynamic viscosity as a function of temperature, density, absolute pressure, and species mass-fraction if applicable.

`double ANYNAME_thermal_conductivity(double T, double Rho, double P, double yi[])`

returns the value of thermal conductivity as a function of temperature, density, absolute pressure, and species mass-fraction if applicable.

`double ANYNAME_rho_t(double T, double Rho, double P, double yi[])`

returns the value of $\frac{d\rho}{dT}$ at constant pressure as a function of temperature, density, absolute pressure, and species mass-fraction if applicable.

`double ANYNAME_rho_p(double T, double Rho, double P, double yi[])`

returns the value of $\frac{d\rho}{dp}$ at constant temperature as a function of temperature, density, absolute pressure, and species mass-fraction if applicable.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

double ANYNAME_enthalpy_t(double T, double Rho, double P, double yi[])

returns the value of $\frac{dh}{dT}$ at constant pressure as a function of temperature, density, absolute pressure, and species mass-fraction if applicable. Note that by definition $dh/\ dt==c_p$, so this function should simply return the specific heat value.

double ANYNAME_enthalpy_p(double T, double Rho, double P, double yi[])

returns the value of $\frac{dh}{dp}$ at constant temperature as a function of temperature, density, absolute pressure, and species mass-fraction if applicable.

double ANYNAME_enthalpy_prime(double T,double Rho,double P,double yi[],double hi[])

returns the value of the mixture enthalpy as a function of temperature, density, absolute pressure, and species mass fraction. In addition, your UDF must set the elements of the double array `hi[]` to the enthalpy of each species, in the same order as they are referenced in the mass fraction array `yi[]`. Note that the enthalpy in the function `enthalpy_prime` is defined as the sum of sensible enthalpy plus species formation enthalpy, and you should make sure that its computation is consistent with the sensible enthalpy function ANYNAME_`enthalpy`. The function ANYNAME_`enthalpy_prime` is required for the calculation of the heat of reactions, if chemical reactions are being simulated. If you are not solving reacting flows, the function ANYNAME_`enthalpy_prime` can simply be omitted.

At the end of the code you must define a structure of type RGAS_`Function` whose members are pointers to the principal functions listed above. The structure is of type RGAS_`Function` and its name is `RealGasFunctionList`.

Important

It is imperative that the sequence of function pointers shown below be followed. Otherwise, your real gas model will not load properly into the ANSYS Fluent code.

```
UDF_EXPORT RGAS_Functions RealGasFunctionList =
{
 ANYNAME_Setup,                  /* Setup initialize */
 ANYNAME_density,                /* density */
 ANYNAME_enthalpy,               /* sensible enthalpy */
 ANYNAME_entropy,                /* entropy */
 ANYNAME_specific_heat,          /* specific_heat */
 ANYNAME_mw,                     /* molecular_weight */
 ANYNAME_speed_of_sound,         /* speed_of_sound */
 ANYNAME_viscosity,              /* viscosity */
 ANYNAME_thermal_conductivity,   /* thermal_conductivity */
 ANYNAME_rho_t,                  /* drho/dT |const p */
 ANYNAME_rho_p,                  /* drho/dp |const T */
 ANYNAME_enthalpy_t,             /* dh/dT |const p */
 ANYNAME_enthalpy_p              /* dh/dp |const T */
 ANYNAME_enthalpy_prime          /* enthalpy */
};
```

If volumetric reactions are not being simulated, then the function ANYNAME_`enthalpy_prime` can be removed or ignored from the `RealGasFunctionList` structure described here.

The principal set of functions described are the only functions in the UDRGM that will be interacting directly with the ANSYS Fluent code. In many cases, your model may require further functions that will be called from the principal function set. For example, when multiple-species real gas model UDFs are written, the principal functions will return the mixture thermodynamic properties based on some specified mixing-law. Therefore, you may want to add further functions that will return the thermodynamic properties for the individual species. These auxiliary functions will be called from the principal set of

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

functions. See User-Defined Real Gas Models (UDRGM) in the Fluent Customization Manual for examples that clearly illustrate this strategy.

7.16.5.3. Compiling Your UDRGM C Functions and Building a Shared Library File

This section presents the steps you must follow to compile your UDRGM C code and build a shared library file. This process requires the use of a C compiler. For more details on compiler requirements, see Compilers in the *Fluent Customization Manual*. To use the UDRGM you must first build the UDRGM library by compiling your UDRGM C code and then load the library into the ANSYS Fluent code. The UDRGM shared library is built in the same way that the ANSYS Fluent executable itself is built. Internally, a script called `Makefile` is used to invoke the system C compiler to build an object code library that contains the native machine language translation of your higher-level C source code. This shared library is then loaded into ANSYS Fluent (either at run time or automatically when a case file is read) by a process called *dynamic loading*. The object libraries are specific to the computer architecture being used, as well as to the particular version of the ANSYS Fluent executable being run. The libraries must, therefore, be rebuilt any time ANSYS Fluent is upgraded, when the computer's operating system level changes, or when the job is run on a different type of computer. The general procedure for compiling UDRGM C code is as follows:

- Place the UDRGM C code in the folder, that is, where your case file resides.
- Launch ANSYS Fluent.
- Read your case file into ANSYS Fluent.
- You can now compile your UDRGM C code and build a shared library file using either the graphical interface or the text command interface.

Important

To build UDRGM library you will use the compiled UDF utilities. However, you will not use the UDF utilities to load the library. A separate loading area for the UDRGM library will be used.

7.16.5.3.1. Compiling the UDRGM Using the Graphical Interface

If the build is successful, then the compiled library will be placed in the appropriate architecture folder (for example, `win64/2d`). By default the library name is `libudf.so` (`libudf.dll` on Windows).

More information on compiled UDFs and building libraries using the ANSYS Fluent graphical user interface can be found in the Fluent Customization Manual.

7.16.5.3.2. Compiling the UDRGM Using the Text Interface

The UDRGM library can be compiled in the text command interface as follows:

- Select the menu item `define` → `user-defined` → `compiled-functions` .
- Select the `compile` option.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Enter the compiled UDF library name.

  Important

  The name given here is the name of the folder where the shared library (for example, `libudf`) will reside. For example, if you press **Enter** then a folder should exist with the name `libudf`, and this folder will contain a library file called `libudf`. If, however, you type a new library name such as `myrealgas`, then a folder called `myrealgas` will be created and it will contain the library `libudf`.

- Continue on with the procedure when prompted.

- Enter the C source file names.

  Important

  Ideally you should place all of your functions into a single file. However, you can split them into separate files if desired.

- Enter the header file names, if applicable. If you do not have an extra header file then press **Enter** when prompted.

ANSYS Fluent will then start compiling the UDRGM C code and put it in the appropriate architecture folder.

Example:

```
 > define/user-defined/compiled-functions
   load OR compile ? [load]  compile

   Compiled UDF library name: ["libudf"] my_lib

    Make sure that UDF source files are in the folder
    that contains your case and data files. If you have
    an existing libudf folder, please remove this
    folder to ensure that latest files are used.
   Continue?[yes] RETURN

   Give C-Source file names:
   First file name: [""] my_c_file.c RETURN

   Next file name: [""] RETURN

   Give header file names:
   First file name: [""] my_header_file.h RETURN
```

7.16.5.3.3. Loading the UDRGM Shared Library File

Load the UDRGM library:

- Go to the following menu item in the text command interface.

  `define` → `user-defined` → `real-gas-models`

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select one of the following
  - `user-defined-real-gas-model` if you are modeling a single-species real gas fluid
  - `user-defined-multispecies-real-gas-model` if you are modeling a multiple-species fluid-mixture
- Turn on the real gas model.
  - For single-species:

```
use user defined real gas? [no]   yes
```

  - For multiple-species:

```
use multispecies user defined real gas? [no]   yes
```

ANSYS Fluent will ask for the location of the user-defined real gas library. You can enter either the name of the folder where the UDRGM shared library is called or the entire path to the UDRGM shared library.

If the loading of the UDRGM library is successful you will see a message similar to the following:

```
Opening user-defined realgas library "RealgasLibraryname"...
Library "RealgasDirName/lnamd64/2d/libudf.so" opened
Setting material "air" to a real-gas...
Loading Real-RealGasPrefexLable Library:
```

7.16.5.4. UDRGM Example: Ideal Gas Equation of State

This section describes an example of a user-defined real gas model. You can use this example as the basis for your own UDRGM code. In this simple example, the standard ideal gas equation of state is used in the UDRGM. See User-Defined Real Gas Models (UDRGM) in the Fluent Customization Manual for more examples of UDRGM functions, including multi-species real gas and reacting real-gas examples.

p = pressure

T = temperature

C_p = specific heat

H = enthalpy

S = entropy

ρ = density

c = speed of sound

R = universal gas constant/molecular weight

The ideal gas equation of state can be written in terms of pressure and temperature as

$$\rho=\frac{p}{RT} \tag{7.119}$$

The specific heat is defined to be constant $C_p = 1006.42$.

The enthalpy is, therefore, defined as

$$H=C_pT \tag{7.120}$$

and entropy is given by

$$S=C_p\log\left(\frac{T}{T_{ref}}\right)+R\log\left(\frac{p_{ref}}{p}\right) \tag{7.121}$$

where T_{ref} = 288.15 K and p_{ref} = 101325 Pa

The speed of sound is simply defined as

$$c=\sqrt{TC_p\frac{R}{\left(C_p-R\right)}} \tag{7.122}$$

The density derivatives are:

$$\left(\frac{d\rho}{dp}\right)_T=\frac{1}{RT} \tag{7.123}$$

$$\left(\frac{d\rho}{dT}\right)_p=-\frac{p}{RT^2}=-\frac{\rho}{T} \tag{7.124}$$

The enthalpy derivatives are:

$$\left(\frac{dH}{dT}\right)_p=C_p \tag{7.125}$$

$$\left(\frac{dH}{dp}\right)_T=\frac{C_p}{\rho R}\left[1-\frac{p}{\rho}\frac{d\rho}{dp}\right]=0 \tag{7.126}$$

When you enable the real gas model and load the library successfully into ANSYS Fluent, you will be using the equation of state and other fluid properties from this library rather than the one built into the ANSYS Fluent code.

7.16.5.4.1. Ideal Gas UDRGM Code Listing

```
/************************************************************************/
/* User Defined Real Gas Model :                                        */
/* For Ideal Gas Equation of State                                      */
/*                                                                      */
/************************************************************************/
#include "udf.h"
#include "stdio.h"
#include "ctype.h"
#include "stdarg.h"

#define MW 28.966   /* molec. wt. for single gas (Kg/Kmol) */
#define RGAS (UNIVERSAL_GAS_CONSTANT/MW)

static int (*usersMessage)(char *,...);
static void (*usersError)(char *,...);


DEFINE_ON_DEMAND(I_do_nothing)
{
   /* This is a dummy function to allow us to use */
   /* the Compiled UDFs utility      */
}


void IDEAL_error(int err, char *f, char *msg)
{
 if (err)
   usersError("IDEAL_error (%d) from function: %s\n%s\n",err,f,msg);
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
}

void IDEAL_Setup(Domain *domain, cxboolean vapor_phase, char *filename,
    int (*messagefunc)(char *format, ...),
    void (*errorfunc)(char *format, ...))
{  /* Use this function for any initialization or model setups*/
     usersMessage = messagefunc;
     usersError  = errorfunc;
     usersMessage("\nLoading Real-Ideal Library: %s\n", filename);
}

double IDEAL_density(cxboolean vapor_phase, double Temp, double press, double yi[])
{
     double r = press/(RGAS*Temp); /* Density at Temp & press */
     return r;      /* (Kg/m^3) */
}

double IDEAL_specific_heat(double Temp, double density, double P, double yi[])
{
     double cp=1006.43;
     return cp;     /* (J/Kg/K) */
}

double IDEAL_enthalpy(double Temp, double density, double P, double yi[])
{
     double h=Temp*IDEAL_specific_heat(Temp, density, P, yi);
     return h;      /* (J/Kg) */
}

#define TDatum 288.15
#define PDatum 1.01325e5

double IDEAL_entropy(double Temp, double density, double P, double yi[])
{
 double s=IDEAL_specific_heat(Temp,density,P,yi)*log(fabs(Temp/TDatum))+
          RGAS*log(fabs(PDatum/P));
    return s;      /* (J/Kg/K) */
}


double IDEAL_mw(double yi[])
{
     return MW;     /* (Kg/Kmol) */
}

double IDEAL_speed_of_sound(double Temp, double density, double P, double yi[])
{
     double cp=IDEAL_specific_heat(Temp,density,P,yi);
     return sqrt(Temp*cp*RGAS/(cp-RGAS)); /* m/s */
}

double IDEAL_viscosity(double Temp, double density, double P, double yi[])
{
     double mu=1.7894e-05;
     return mu;     /* (Kg/m/s) */
}

double IDEAL_thermal_conductivity(double Temp, double density, double P,
              double yi[])
{
     double ktc=0.0242;
     return ktc;      /* W/m/K */
}
double IDEAL_rho_t(double Temp, double density, double P, double yi[])
{
     /* derivative of rho wrt. Temp at constant p */
     double rho_t=-density/Temp;
     return rho_t;     /* (Kg/m^3/K) */
}

double IDEAL_rho_p(double Temp, double density, double P, double yi[])
{
     /* derivative of rho wrt. pressure at constant T */
```

```
     double rho_p=1.0/(RGAS*Temp);
     return rho_p;    /* (Kg/m^3/Pa) */
}

double IDEAL_enthalpy_t(double Temp, double density, double P, double yi[])
{
     /* derivative of enthalpy wrt. Temp at constant p */
     return IDEAL_specific_heat(Temp, density, P, yi);
}

double IDEAL_enthalpy_p(double Temp, double density, double P, double yi[])
{
     /* derivative of enthalpy wrt. pressure at constant T  */
     /* general form dh/dp|T = (1/rho)*[ 1 + (T/rho)*drho/dT|p] */
     /* but for ideal gas dh/dp = 0        */
     return 0.0 ;
}

UDF_EXPORT RGAS_Functions RealGasFunctionList =
{
     IDEAL_Setup,                   /* initialize */
     IDEAL_density,                 /* density */
     IDEAL_enthalpy,                /* enthalpy */
     IDEAL_entropy,                 /* entropy */
     IDEAL_specific_heat,           /* specific_heat */
     IDEAL_mw,                      /* molecular_weight */
     IDEAL_speed_of_sound,          /* speed_of_sound */
     IDEAL_viscosity,               /* viscosity */
     IDEAL_thermal_conductivity,    /* thermal_conductivity */
     IDEAL_rho_t,                   /* drho/dT |const p */
     IDEAL_rho_p,                   /* drho/dp |const T */
     IDEAL_enthalpy_t,              /* dh/dT |const p  */
     IDEAL_enthalpy_p               /* dh/dp |const T  */
};
/************************************************************/
```

7.16.5.5. Additional UDRGM Examples

You can find the following additional UDRGM examples in the Fluent Customization Manual:

- The Aungier Redlich Kwong equation of state for single component flow. See UDRGM Example: Redlich-Kwong Equation of State in the Fluent Customization Manual for details.
- A simple example of a multi-species real-gas model. See UDRGM Example: Multiple-Species Real Gas Model in the Fluent Customization Manual for details.
- A real gas model example with the Aungier Redlich Kwong equation of state, ideal gas mixing rules and volumetric reactions. See UDRGM Example: Real Gas Model with Volumetric Reactions in the Fluent Customization Manual for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 8: Modeling Basic Fluid Flow

This chapter describes the basic physical models that ANSYS Fluent provides for fluid flow and the commands for defining and using them. Models for flows in moving zones (including sliding and dynamic meshes) are explained in Modeling Flows with Moving Reference Frames (p. 603), models for turbulence are described in Modeling Turbulence (p. 749), and models for heat transfer (including radiation) are presented in Modeling Heat Transfer (p. 825). An overview of modeling species transport and reacting flows is provided in Modeling Species Transport and Finite-Rate Chemistry (p. 957), details about models for species transport and reacting flows are described in Modeling Species Transport and Finite-Rate Chemistry (p. 957) – Modeling a Composition PDF Transport Problem (p. 1113), and models for pollutant formation are presented in Modeling Pollutant Formation (p. 1153). The discrete phase model is described in Modeling Discrete Phase (p. 1235), general multiphase models are described in Modeling Multiphase Flows (p. 1365), and the melting and solidification model is described in Modeling Solidification and Melting (p. 1527). For information on modeling porous media, porous jumps, and lumped parameter fans and radiators, see Cell Zone and Boundary Conditions (p. 237).

The information in this chapter is presented in the following sections:

8.1. User-Defined Scalar (UDS) Transport Equations
8.2. Periodic Flows
8.3. Swirling and Rotating Flows
8.4. Compressible Flows
8.5. Inviscid Flows

8.1. User-Defined Scalar (UDS) Transport Equations

For additional information, see the following sections:

8.1.1. Introduction
8.1.2. UDS Theory
8.1.3. Setting Up UDS Equations in ANSYS Fluent

8.1.1. Introduction

ANSYS Fluent can solve the transport equation for an arbitrary, user-defined scalar (UDS) in the same way that it solves the transport equation for a scalar such as species mass fraction. Extra scalar transport equations may be needed in certain types of combustion applications or for example in plasma-enhanced surface reaction modeling. ANSYS Fluent allows you to define additional scalar transport equations in your model in the User-Defined Scalars Dialog Box (p. 2829).

8.1.2. UDS Theory

UDS theory is described in the following sections:

8.1.2.1. Single Phase Flow
8.1.2.2. Multiphase Flow

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.1.2.1. Single Phase Flow

For an arbitrary scalar ϕ_k, ANSYS Fluent solves the equation

$$\frac{\partial \rho \phi_k}{\partial t} + \frac{\partial}{\partial x_i}\left(\rho u_i \phi_k - \Gamma_k \frac{\partial \phi_k}{\partial x_i}\right) = S_{\phi_k} \quad k = 1, \ldots, N \tag{8.1}$$

where Γ_k and S_{ϕ_k} are the diffusion coefficient and source term supplied by you for each of the N scalar equations. Note that Γ_k is defined as a tensor in the case of anisotropic diffusivity. The diffusion term is therefore $\nabla \cdot \left(\Gamma_k \cdot \phi_k\right)$

For isotropic diffusivity, Γ_k could be written as $\Gamma_k I$ where I is the identity matrix.

For the steady-state case, ANSYS Fluent will solve one of the three following equations, depending on the method used to compute the convective flux:

- If convective flux is *not* to be computed, ANSYS Fluent will solve the equation

$$-\frac{\partial}{\partial x_i}\left(\Gamma_k \frac{\partial \phi_k}{\partial x_i}\right) = S_{\phi_k} \quad k = 1, \ldots, N \tag{8.2}$$

  where Γ_k and S_{ϕ_k} are the diffusion coefficient and source term supplied by you for each of the N scalar equations.

- If convective flux is to be computed with mass flow rate, ANSYS Fluent will solve the equation

$$\frac{\partial}{\partial x_i}\left(\rho u_i \phi_k - \Gamma_k \frac{\partial \phi_k}{\partial x_i}\right) = S_{\phi_k} \quad k = 1, \ldots, N \tag{8.3}$$

- It is also possible to specify a user-defined function to be used in the computation of convective flux. In this case, the user-defined mass flux is assumed to be of the form

$$F = \int_S \rho \vec{u} \cdot d\vec{S} \tag{8.4}$$

where $d\vec{S}$ is the face vector area.

Important

User-defined scalars in solid zones do not take into account the convective term with moving reference frames.

8.1.2.2. Multiphase Flow

For multiphase flows, ANSYS Fluent solves transport equations for two types of scalars: *per phase* and *mixture*. For an arbitrary k scalar in *phase-l*, denoted by ϕ_l^k, ANSYS Fluent solves the transport equation inside the volume occupied by *phase-l*

$$\frac{\partial \alpha_l \rho_l \phi_l^k}{\partial t} + \nabla \cdot \left(\alpha_l \rho_l \vec{u}_l \phi_l^k - \alpha_l \Gamma_l^k \nabla \phi_l^k\right) = S_l^k \quad k = 1, \ldots, N \tag{8.5}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where α_l, ρ_l, and $\vec{u}_l$ are the volume fraction, physical density, and velocity of *phase-l*, respectively. Γ_l^k and S_l^k are the diffusion coefficient and source term, respectively, which you will need to specify. In this case, scalar ϕ_l^k is associated only with one phase (*phase-l*) and is considered an individual field variable of *phase-l*.

The mass flux for *phase-l* is defined as

$$F_l = \int_S \alpha_l \rho_l \vec{u}_l \cdot d\vec{S} \tag{8.6}$$

If the transport variable described by scalar ϕ_l^k represents the physical field that is shared between phases, or is considered the same for each phase, then you should consider this scalar as being associated with a mixture of phases, ϕ^k. In this case, the generic transport equation for the scalar is

$$\frac{\partial \rho_m \phi^k}{\partial t} + \nabla \cdot \left(\rho_m \vec{u}_m \phi^k - \Gamma_m^k \nabla \phi^k \right) = S^{k_m} \; k = 1, \ldots, N \tag{8.7}$$

where mixture density ρ_m, mixture velocity $\vec{u}_m$, and mixture diffusivity for the scalar k Γ_m^k are calculated according to

$$\rho_m = \sum_l \alpha_l \rho_l \tag{8.8}$$

$$\rho_m \vec{u}_m = \sum_l \alpha_l \rho_l \vec{u}_l \tag{8.9}$$

$$F_m = \int_S \rho_m \vec{u}_m \cdot d\vec{S} \tag{8.10}$$

$$\Gamma_m^k = \sum_l \alpha_l \Gamma_l^k \tag{8.11}$$

$$S_m^k = \sum_l S_l^k \tag{8.12}$$

To calculate mixture diffusivity, you must specify individual diffusivities for each material associated with individual phases.

Note that if the user-defined mass flux option is activated, then mass fluxes shown in Equation 8.6 (p. 575) and Equation 8.10 (p. 575) must be replaced in the corresponding scalar transport equations. For more information about the theoretical background of user-defined scalar transport equations, see User-Defined Scalar (UDS) Transport Equations in the Theory Guide.

8.1.3. Setting Up UDS Equations in ANSYS Fluent

ANSYS Fluent allows you to define up to 50 user-defined scalar (UDS) transport equations in your model. The general scalar transport equation, Equation 1.8 in the Theory Guide, is shown below with the four terms (transient, flux, diffusivity, source) that you can customize. (Equation 8.13 (p. 575)). You will define a UDS transport equation by setting the parameters for these four terms.

$$\underbrace{\frac{\partial \rho \phi_k}{\partial t}}_{unsteady} + \frac{\partial}{\partial x_i} \left(\underbrace{F_i \phi_k}_{convection} - \Gamma_k \underbrace{\frac{\partial \phi_k}{\partial x_i}}_{diffusion} \right) = \underbrace{S_{\phi_k}}_{sources} \qquad k = 1, \ldots, N_{scalars} \tag{8.13}$$

In addition, you can set boundary conditions for the variables within cells of a fluid or solid zone for a particular scalar equation. This is done by fixing the value of ϕ_k in Equation 8.13 (p. 575). When ϕ_k is fixed in a given cell, the UDS scalar transport is not solved and the cell is not included when the residual sum is computed. Additionally, you can also specify custom boundary conditions in the mixture on all wall, inflow, and outflow boundaries on a per-scalar basis.

The procedures for setting up a user-defined scalar (UDS) equation for single-phase and multiphase flows are outlined below. Note that a significant difference between a UDS for a single-phase versus a multiphase application is that you must associate each UDS with its corresponding phase domain or mixture domain, depending on your application. If you supply UDFs for transient terms, convective fluxes, and sources, you must be aware that they are directly called from the phase or mixture domains, according to the scalar association settings.

When setting up a UDS, ensure that the scalar transport problem is well defined with proper Dirichlet/Neumann boundary conditions. Depending on the nature of the problem, the UDS equation may need to converge further than the default convergence criterion of 0.1. Set the convergence criterion according to your needs.

In some instances, the residuals may not reflect the quality of the solution. In these cases you should create monitors on the UDS and use these monitors to judge solution convergence. There are several controls that you can adjust to improve convergence, however some of these adjustments can increase the computation time. The following controls can help with convergence:

- Using a fixed cycle, such as **F-Cycle** in place of **Flexible** cycle in the **Advanced Solution Controls** dialog box.
- Using an additional stabilizer such as the Biconjugate gradient stabilization method.
- Using an ILU smoother.
- Increasing the pre- and post-sweeps available in the **Advanced Solution Controls** dialog box.
- Adjusting the under relaxation factors.

See the Fluent Customization Manual for information on using UDFs to define scalar quantities.

8.1.3.1. Single Phase Flow

1. Specify the number of UDS equations you require in the User-Defined Scalars Dialog Box (p. 2829) (Figure 8.1: The User-Defined Scalars Dialog Box (p. 577)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parameters & Customization → User Defined Scalars New...

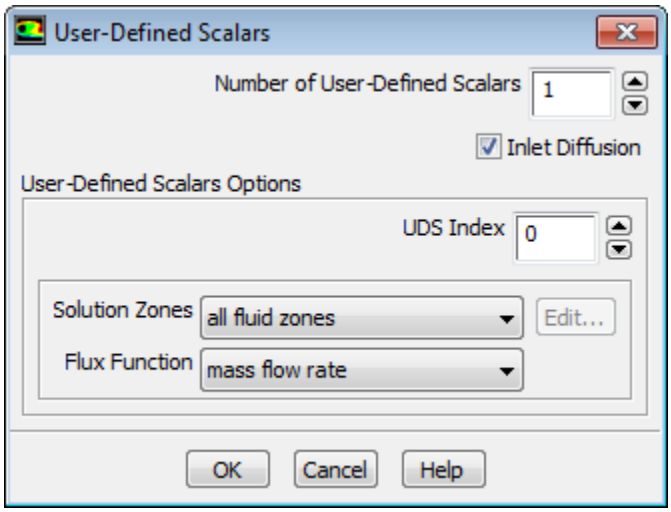
Important

The maximum number of user-defined scalar transport equations you can define is 50. ANSYS Fluent assigns numbers to the equations starting with 0.

Important

Note that ANSYS Fluent assigns a default name for each scalar equation (`User Scalar 0`, `User Scalar 1`, and so on). These labels will appear in graphics dialog boxes in ANSYS Fluent. You can change them by means of a UDF. See the Fluent Customization Manual for details.

Figure 8.1: The User-Defined Scalars Dialog Box



2. Enable **Inlet Diffusion** if you want to include the diffusion term in the UDS transport equation for all inflow and outflow boundaries.

3. Set the first user-defined scalar equation parameters by making sure that the **UDS Index** is set to 0.

   a. Specify the **Solution Zones** you want the scalar equation to be solved in as **all fluid zones**, **all solid zones**, **all zones** (fluid and solid) or **selected zones**. If you choose **selected zones**, click the **Edit** button to view the list of zones you can select.

   b. Specify the **Flux Function** to be **none**, **mass flow rate**, or a user-defined function (UDF). The **Flux Function** determines how the convective flux is computed, which determines the equation that ANSYS Fluent solves for the user-defined scalar. Selecting **none**, **mass flow rate**, or a user-defined function results in ANSYS Fluent solving Equation 1.9, Equation 1.10, or Equation 1.11, respectively (in the Theory Guide). See the Fluent Customization Manual for details on flux UDFs.

   c. Specify the **Unsteady Function** to be **none**, **default**, or a user-defined function (UDF). Select **none** for a steady state solution and **default** if you want the transient term in Equation 1.8 in the Theory Guide). See the separate Fluent Customization Manual for details on unsteady UDFs.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

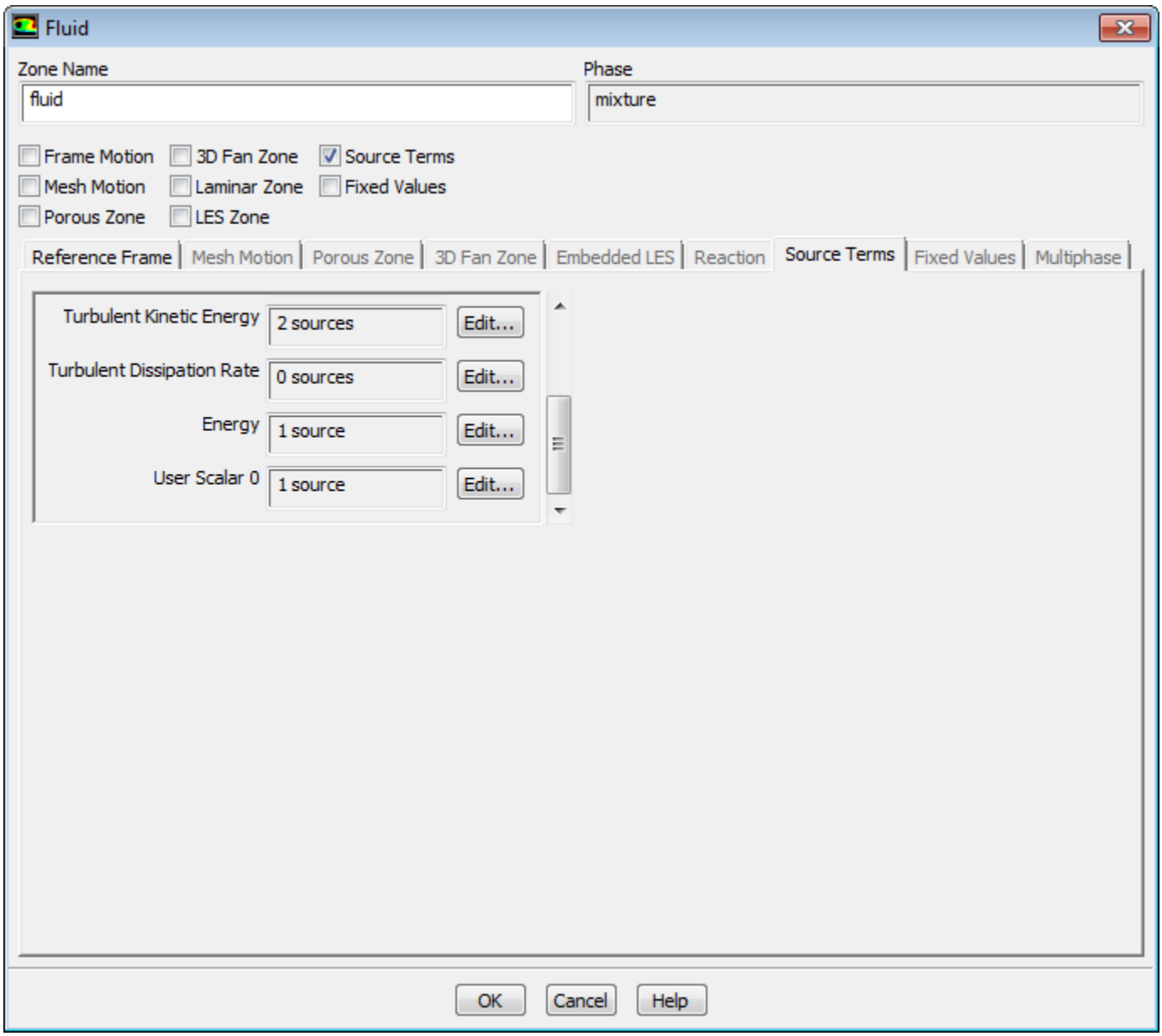
d. Repeat this process for each scalar equation by incrementing the **UDS Index**.

e. Click **OK** when all user scalar equations have been defined.

4. To specify source term(s) for each of the N UDS equations, enable the **Source Terms** option in the **Fluid** or **Solid** dialog box (Figure 8.2: The Fluid Dialog Box with Inputs for Source Terms for a User-Defined Scalar (p. 578)) and click the **Source Terms** tab. The source parameters will be displayed.

Setup → Cell Zone Conditions

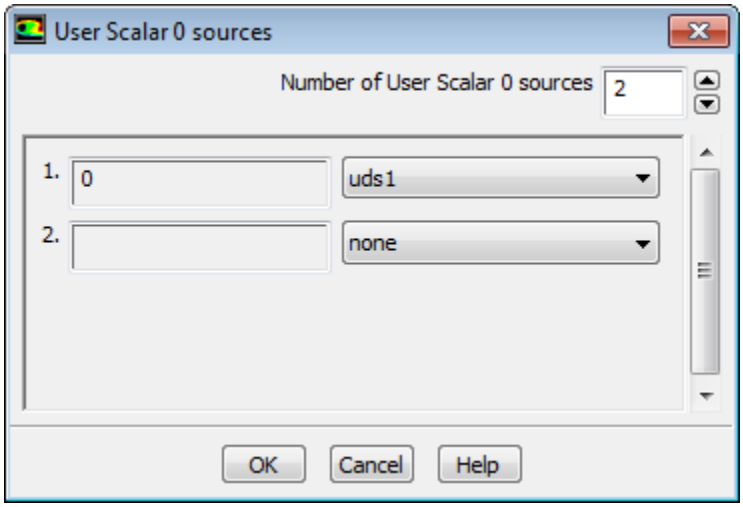
Figure 8.2: The Fluid Dialog Box with Inputs for Source Terms for a User-Defined Scalar



a. Specify the number of sources you require for each scalar equation by clicking on the **Edit...** button next to the scalar name (for example, **User Scalar 0**). This will open the **User Scalar 0 sources** dialog box (Figure 8.3: The User Scalar Sources Dialog Box (p. 579)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.3: The User Scalar Sources Dialog Box

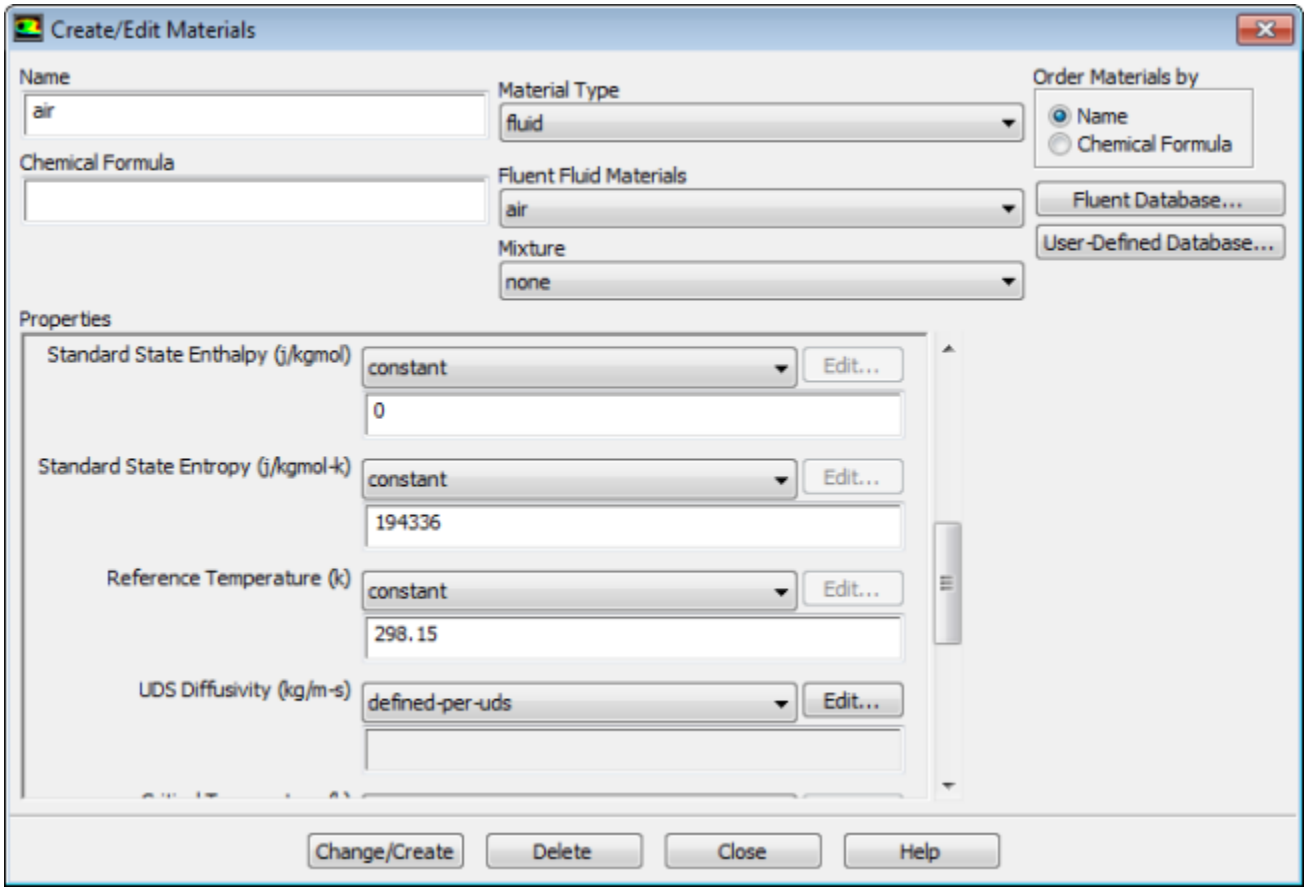


b. Specify the **Number of User Scalar Sources** for the scalar equation by incrementing the counter. Based on the value you have chosen, the sources will be added to the list in the dialog box. Specify each source to be **none**, **constant**, or a user-defined function (UDF). For details on defining a UDF scalar source, see the Fluent Customization Manual. Click **OK** when you have specified all scalar sources.

5. To specify diffusivity for each of the N UDS equations, display the Materials Task Page (p. 2300) (Figure 8.4: The Materials Dialog Box with Input for Diffusivity for UDS Equations (p. 580)) and select either **defined-per-uds** (the default) or **user-defined** in the drop-down list for **UDS Diffusivity**.

Setup → **Materials** → **Create/Edit**

Figure 8.4: The Materials Dialog Box with Input for Diffusivity for UDS Equations



See User-Defined Scalar (UDS) Diffusivity (p. 508) for details on the different options available to you for defining diffusion coefficients.

6. To specify boundary conditions for the user-defined scalars on wall, inflow, and outflow boundaries, you can define a specific value or a specific flux for each scalar. A coupled boundary condition can be specified on two-sided walls for scalars that are to be solved in regions on both sides of the wall (that is, scalars solved in both **fluid and solid** zones).

   Setup → Boundary Conditions

   a. In the **UDS** tab under **User Defined Scalar Boundary Condition**, select either **Specified Flux** or **Specified Value** in the drop-down list next to each scalar (for example, **User Scalar 0**) for a boundary wall. For interior walls, select **Coupled Boundary** if the scalars are to be solved on both sides of a two-sided wall. Note that the **Coupled Boundary** option will only show up in the drop-down list if the scalar is defined in the **fluid and solid** zones in the User-Defined Scalars Dialog Box (p. 2829).

   b. Under **User Defined Scalar Boundary Value**, enter a constant value or select a user-defined function from the drop-down list for each scalar. If you select **Specified Flux**, your input will be the value of the flux at the boundary (that is, the negative of the term in parenthesis on the left-hand side of in the Theory Guide) dot [as in the dot product of] n [as in the vector, n], where n is the normal into the domain). If you select **Specified Value**, your input will be the value of the scalar itself at the boundary. See the Fluent Customization Manual for information on using UDFs for UDS boundary conditions.

7. Set the solution parameters in the **Solution Controls** task page, specify an initial value for each UDS (as you do for all other scalar transport equations), and calculate a solution.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Examine the results using the usual postprocessing tools. In each postprocessing dialog box, the list of field variables will include the **User Defined Scalars...** category, which contains the value of each UDS and its diffusion coefficient (Γ_k in Equation 1.8, Equation 1.9, Equation 1.10, or Equation 1.11 (in the Theory Guide):

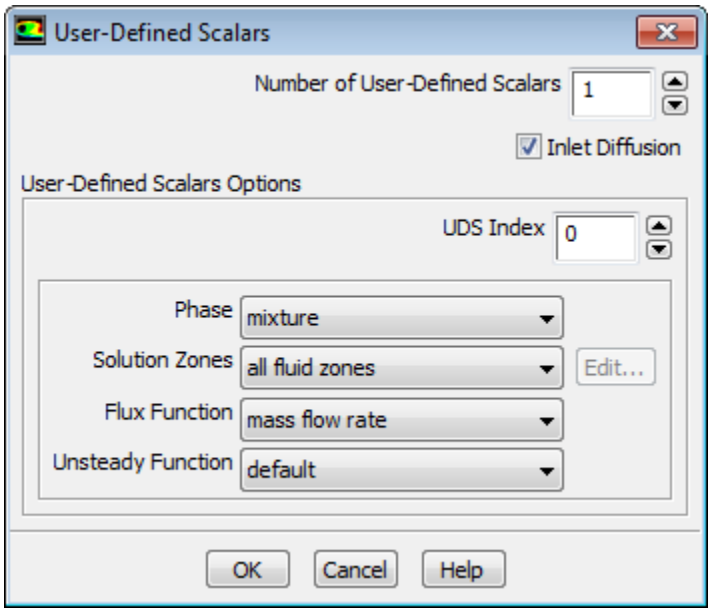
   - **User Scalar-n**
   - **Diffusion Coef. of Scalar-n**

8.1.3.2. Multiphase Flow

1. Specify the number of scalars in the User-Defined Scalars Dialog Box (p. 2829) (Figure 8.5: The User-Defined Scalars Dialog Box for a Multiphase Flow (p. 581)).

   Parameters & Customization → **User Defined Scalars** **New...**

   Figure 8.5: The User-Defined Scalars Dialog Box for a Multiphase Flow



Important

The maximum number of user-defined scalar transport equations you can define is 50. ANSYS Fluent assigns numbers to the equations starting with 0. The default association type is set to `mixture` for all scalars.

Important

Note that ANSYS Fluent assigns a default name for each scalar equation (`User Scalar 0`, `User Scalar 1`, and so on). These labels will appear in graphics dialog boxes in ANSYS Fluent. You can change them by means of a UDF. See the Fluent Customization Manual for details.

2. Keep the default **Inlet Diffusion** enabled if you want to include the diffusion term in the UDS transport equation for all inflow and outflow boundaries.

3. Set the first user-defined scalar equation parameters by making sure that the **UDS Index** is set to 0.

   a. Select the **Phase** you want the scalar equation solved in as a primary phase, secondary phase, or the mixture.

   b. Specify the **Solution Zones** you want the scalar equation to be solved in as **all fluid zones**, **all solid zones**, **all zones** (fluid and solid) or **selected zones**. If you choose **selected zones**, click the **Edit** button to view the list of zones you can select.

   c. Specify the **Flux Function** to **Unsteady Function** the same way as you would for a single phase flow (see above).

   d. Repeat this process for each scalar equation by incrementing the **UDS Index**.

   e. Click **OK** when all user scalar equations have been defined.

4. Specify source term(s) for each of the N UDS equations in the **Fluid** or **Solid** dialog box as described for a single phase flow (see above).

5. Specify boundary conditions for the user-defined scalars in the mixture on all wall, inflow, and outflow boundary as described for a single phase flow (see above).

6. Set the solution parameters, specify an initial value for each UDS (as you do for all other scalar transport equations), and calculate a solution.

8.2. Periodic Flows

Periodic flow occurs when the physical geometry of interest and the expected pattern of the flow/thermal solution have a periodically repeating nature. Two types of periodic flow can be modeled in ANSYS Fluent. In the first type, no pressure drop occurs across the periodic planes. In the second type, a pressure drop occurs across translationally periodic boundaries, resulting in "fully-developed" or "streamwise-periodic" flow.

This section discusses streamwise-periodic flow. A description of no-pressure-drop periodic flow is provided in Periodic Boundary Conditions (p. 386), and a description of streamwise-periodic heat transfer is provided in Modeling Periodic Heat Transfer (p. 912).

Information about streamwise-periodic flow is presented in the following sections:
8.2.1. Overview and Limitations
8.2.2. User Inputs for the Pressure-Based Solver
8.2.3. User Inputs for the Density-Based Solvers
8.2.4. Monitoring the Value of the Pressure Gradient
8.2.5. Postprocessing for Streamwise-Periodic Flows

For more information about the theoretical background of periodic flows, see Periodic Flows in the Theory Guide.

8.2.1. Overview and Limitations

More information about periodic flows is presented in the following sections:
8.2.1.1. Overview

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

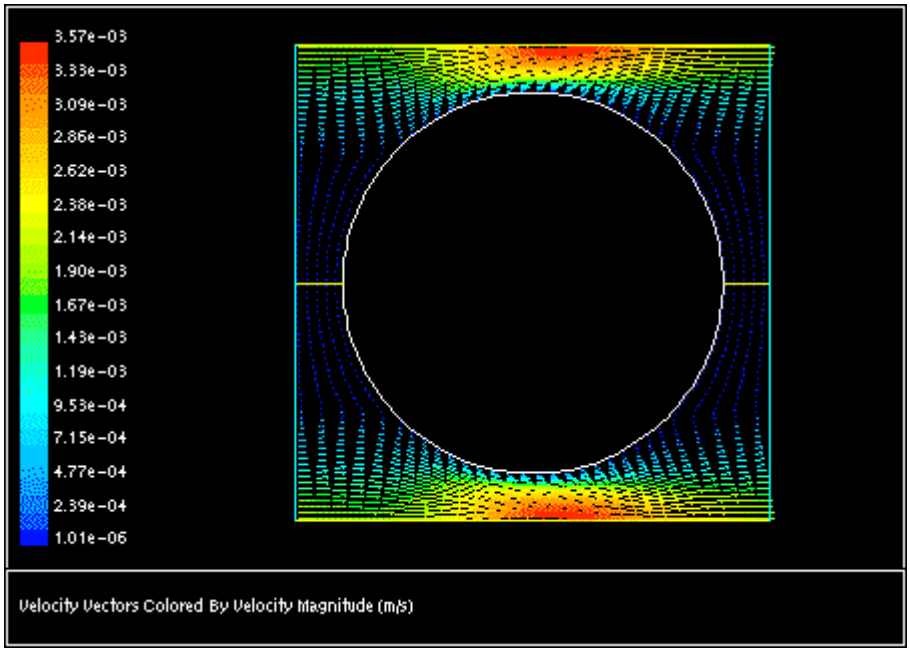
8.2.1.2. Limitations for Modeling Streamwise-Periodic Flow

8.2.1.1. Overview

ANSYS Fluent provides the ability to calculate streamwise-periodic—or "fully-developed"—fluid flow. These flows are encountered in a variety of applications, including flows in compact heat exchanger channels and flows across tube banks. In such flow configurations, the geometry varies in a repeating manner along the direction of the flow, leading to a periodic fully-developed flow regime in which the flow pattern repeats in successive cycles. Other examples of streamwise-periodic flows include fully-developed flow in pipes and ducts. These periodic conditions are achieved after a sufficient entrance length, which depends on the flow Reynolds number and geometric configuration.

Streamwise-periodic flow conditions exist when the flow pattern repeats over some length L, with a constant pressure drop across each repeating module along the streamwise direction. Figure 8.6: Example of Periodic Flow in a 2D Heat Exchanger Geometry (p. 583) depicts one example of a periodically repeating flow of this type that has been modeled by including a single representative module.

Figure 8.6: Example of Periodic Flow in a 2D Heat Exchanger Geometry



8.2.1.2. Limitations for Modeling Streamwise-Periodic Flow

The following limitations apply to modeling streamwise-periodic flow:

- The flow must be incompressible.
- When performing unsteady-state simulations with translational periodic boundary conditions, the specified pressure gradient is recommended.
- If one of the density-based solvers is used, you can specify only the pressure jump; for the pressure-based solver, you can specify either the pressure jump or the mass flow rate.
- No net mass addition through inlets/exits or extra source terms is allowed.
- Species can be modeled only if inlets/exits (without net mass addition) are included in the problem. Reacting flows are not permitted.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Steady particle tracks can be modeled only if the particles have a possibility to leave the domain without generating incomplete trajectories.
- While Eulerian multiphase can be modeled with translational periodic boundary conditions, you cannot use the mass flow rate specification method. However, you can specify a constant pressure gradient.

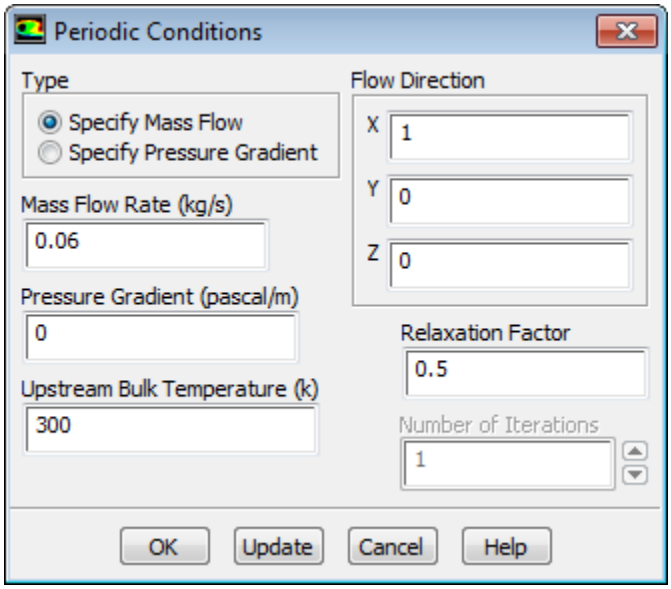
8.2.2. User Inputs for the Pressure-Based Solver

If you are using the pressure-based solver, in order to calculate a spatially periodic flow field with a specified mass flow rate or pressure derivative, you must first create a mesh with translationally periodic boundaries that are parallel to each other and equal in size. You can specify translational periodicity in the Periodic Conditions Dialog Box (p. 2468), as described in Periodic Boundary Conditions (p. 386). (If you need to create periodic boundaries, see Creating Conformal Periodic Zones (p. 214)).

In the Periodic Conditions Dialog Box (p. 2468) that is opened from the Boundary Conditions Task Page (p. 2389), you will complete the following inputs after the mesh has been read into ANSYS Fluent (Figure 8.7: The Periodic Conditions Dialog Box (p. 584)):

Setup → Boundary Conditions → Periodic Conditions...

Figure 8.7: The Periodic Conditions Dialog Box



1. Select either the specified mass flow rate (**Specify Mass Flow**) option or the specified pressure gradient (**Specify Pressure Gradient**) option. For most problems, the mass flow rate across the periodic boundary will be a known quantity; for others, the mass flow rate will be unknown, but the pressure gradient (β in Equation 1.22, in the Theory Guide) will be a known quantity.
2. Specify the mass flow rate and/or the pressure gradient (β in Equation 1.22, in the Theory Guide):
   - If you selected the **Specify Mass Flow** option, enter the desired value for the **Mass Flow Rate**. You can also specify an initial guess for the **Pressure Gradient**, but this is not required.

> **Important**
>
> For axisymmetric problems, the mass flow rate is per 2π radians.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you selected the **Specify Pressure Gradient** option, enter the desired value for **Pressure Gradient**.

3. Define the flow direction by setting the **X,Y,Z** (or **X,Y** in 2D) point under **Flow Direction**. The flow will move in the direction of the vector pointing from the origin to the specified point. The direction vector must be parallel to the periodic translation direction or its opposite.

4. If you chose in step 1 to specify the mass flow rate, set the parameters used for the calculation of β. These parameters are described in detail below.

After completing these inputs, you can solve the periodic velocity field to convergence.

8.2.2.1. Setting Parameters for the Calculation of β

If you choose to specify the mass flow rate, ANSYS Fluent must calculate the appropriate value of the pressure gradient β. You can control this calculation by specifying the **Relaxation Factor** and the **Number of Iterations**, and by supplying an initial guess for β. All of these inputs are entered in the Periodic Conditions Dialog Box (p. 2468).

The **Number of Iterations** sets the number of sub-iterations performed on the correction of β in the pressure correction equation. Because the value of β is not known a priori, it must be iterated on until the **Mass Flow Rate** that you have defined is achieved in the computational model. This correction of β occurs in the pressure correction step of the SIMPLE, SIMPLEC, or PISO algorithm. A correction to the current value of β is calculated based on the difference between the desired mass flow rate and the actual one. The sub-iterations referred to here are performed within the pressure correction step to improve the correction for β before the pressure correction equation is solved for the resulting pressure (and velocity) correction values. The default value of 2 sub-iterations should suffice in most problems, but can be increased to help speed convergence. The **Relaxation Factor** is an under-relaxation factor that controls convergence of this iteration process.

You can also speed up convergence of the periodic calculation by supplying an initial guess for β in the **Pressure Gradient** field. Note that the current value of β will be displayed in this field if you have performed any calculations. To update the **Pressure Gradient** field with the current value at any time, click the **Update** button.

8.2.3. User Inputs for the Density-Based Solvers

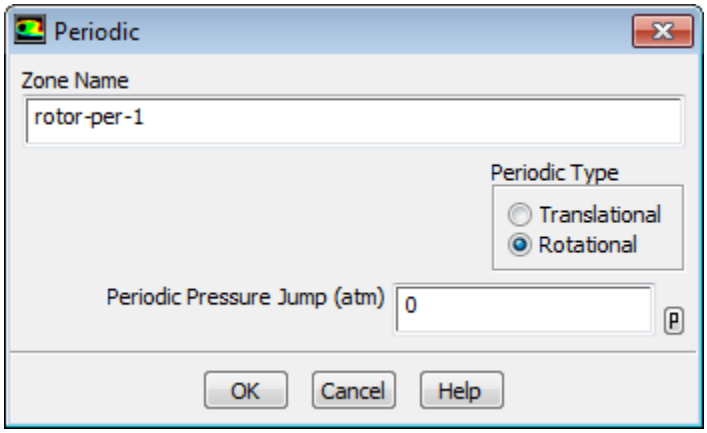
If you are using one of the density-based solvers, in order to calculate a spatially periodic flow field with a specified pressure jump, you must first create a mesh with translationally periodic boundaries that are parallel to each other and equal in size. (If you need to create periodic boundaries, see Creating Conformal Periodic Zones (p. 214).)

Then, follow the steps below:

1. In the Periodic Dialog Box (p. 2427) (Figure 8.8: The Periodic Dialog Box (p. 586)), which is opened from the **Boundary Conditions** task page, indicate that the periodicity is **Translational** (the default).

   Setup → **Boundary Conditions** → **Periodic** → **Edit...**

Figure 8.8: The Periodic Dialog Box



2. Also in the Periodic Dialog Box (p. 2427), set the **Periodic Pressure Jump** (Δp in Equation 1.21 in the Theory Guide).

After completing these inputs, you can solve the periodic velocity field to convergence.

8.2.4. Monitoring the Value of the Pressure Gradient

If you have specified the mass flow rate, you can monitor the value of the pressure gradient β during the calculation by creating a report plot that includes periodic pressure gradient (**per-pr-grad**), to ensure that you reach a converged solution. See Monitoring Statistics (p. 1633) for additional information.

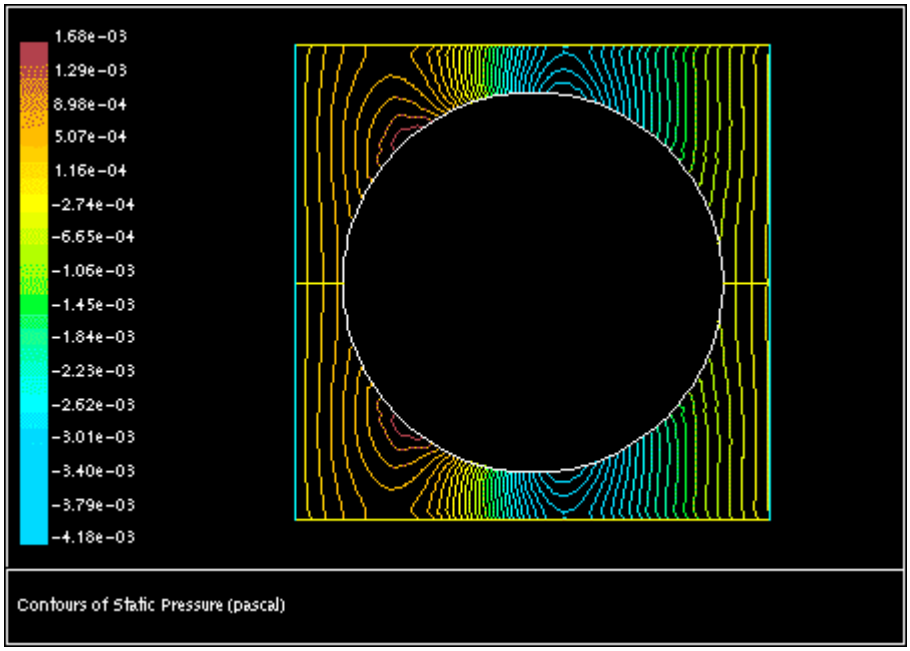
8.2.5. Postprocessing for Streamwise-Periodic Flows

For streamwise-periodic flows, the velocity field should be completely periodic. If a density-based solver is used to compute the periodic flow, the pressure field reported will be the actual pressure p (which is not periodic). If the pressure-based solver is used, the pressure field reported will be the periodic pressure field $\tilde{p}(\vec{r})$ of Equation 1.22, in the Theory Guide. Figure 8.9: Periodic Pressure Field Predicted for Flow in a 2D Heat Exchanger Geometry (p. 587) displays the periodic pressure field in the geometry of Figure 8.6: Example of Periodic Flow in a 2D Heat Exchanger Geometry (p. 583).

If you specified a mass flow rate and had ANSYS Fluent calculate the pressure gradient, you can check the pressure gradient in the streamwise direction (β) by looking at the current value for **Pressure Gradient** in the Periodic Conditions Dialog Box (p. 2468).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.9: Periodic Pressure Field Predicted for Flow in a 2D Heat Exchanger Geometry



8.3. Swirling and Rotating Flows

Many important engineering flows involve swirl or rotation and ANSYS Fluent is well-equipped to model such flows. Swirling flows are common in combustion, with swirl introduced in burners and combustors in order to increase residence time and stabilize the flow pattern. Rotating flows are also encountered in turbomachinery, mixing tanks, and a variety of other applications.

Information about rotating and swirling flows is provided in the following subsections:

8.3.1. Overview of Swirling and Rotating Flows
8.3.2. Turbulence Modeling in Swirling Flows
8.3.3. Mesh Setup for Swirling and Rotating Flows
8.3.4. Modeling Axisymmetric Flows with Swirl or Rotation

For more information about the theoretical background of swirling and rotating flows, see Swirling and Rotating Flows in the Theory Guide.

When you begin the analysis of a rotating or swirling flow, it is essential that you classify your problem into one of the following five categories of flow:

- axisymmetric flows with swirl or rotation
- fully three-dimensional swirling or rotating flows
- flows requiring a moving reference frame
- flows requiring multiple moving reference frames or mixing planes
- flows requiring sliding meshes

Modeling and solution procedures for the first two categories are presented in this section. The remaining three, which all involve "moving zones", are discussed in Modeling Flows with Moving Reference Frames (p. 603).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.3.1. Overview of Swirling and Rotating Flows

An overview of swirling and rotating flows is presented in the following sections:

8.3.1.1. Axisymmetric Flows with Swirl or Rotation
8.3.1.2. Three-Dimensional Swirling Flows
8.3.1.3. Flows Requiring a Moving Reference Frame

8.3.1.1. Axisymmetric Flows with Swirl or Rotation

Your problem may be axisymmetric with respect to geometry and flow conditions but still include swirl or rotation. In this case, you can model the flow in 2D (that is, solve the axisymmetric problem) and include the prediction of the circumferential (or swirl) velocity. It is important to note that while the assumption of axisymmetry implies that there are no circumferential gradients in the flow, there may still be nonzero swirl velocities.

8.3.1.1.1. Momentum Conservation Equation for Swirl Velocity

The tangential momentum equation for 2D swirling flows may be written as

$$\frac{\partial}{\partial t}\left(\rho w\right)+\frac{1}{r}\frac{\partial}{\partial x}\left(r\rho uw\right)+\frac{1}{r}\frac{\partial}{\partial r}\left(r\rho vw\right) = \frac{1}{r}\frac{\partial}{\partial x}\left[r\mu\frac{\partial w}{\partial x}\right] + \frac{1}{r^2}\frac{\partial}{\partial r}\left[r^3\mu\frac{\partial}{\partial r}\left(\frac{w}{r}\right)\right]-\rho\frac{vw}{r} \tag{8.14}$$

where x is the axial coordinate, r is the radial coordinate, u is the axial velocity, v is the radial velocity, and w is the swirl velocity.

8.3.1.2. Three-Dimensional Swirling Flows

When there are geometric changes and/or flow gradients in the circumferential direction, your swirling flow prediction requires a three-dimensional model. If you are planning a 3D ANSYS Fluent model that includes swirl or rotation, you should be aware of the setup constraints listed in Coordinate System Restrictions (p. 589). In addition, you might consider simplifications to the problem that might reduce it to an equivalent axisymmetric problem, especially for your initial modeling effort. Because of the complexity of swirling flows, an initial 2D study, in which you can quickly determine the effects of various modeling and design choices, can be very beneficial.

Important

For 3D problems involving swirl or rotation, there are no special inputs required during the problem setup and no special solution procedures. Note, however, that you may want to use the cylindrical coordinate system for defining velocity-inlet boundary condition inputs, as described in Defining the Velocity (p. 318). Also, you may find the gradual increase of the rotational speed (set as a wall or inlet boundary condition) helpful during the solution process. This is described for axisymmetric swirling flows in Improving Solution Stability by Gradually Increasing the Rotational or Swirl Speed (p. 592).

8.3.1.3. Flows Requiring a Moving Reference Frame

If your flow involves a rotating boundary that moves through the fluid (for example, an impeller blade or a grooved or notched surface), you must use a moving reference frame to model the problem. Such applications are described in detail in Introduction (p. 603). If you have more than one rotating boundary (for example, several impellers in a row), you can use multiple reference frames (described in The Multiple Reference Frame Model (p. 613)) or mixing planes (described in The Mixing Plane Model (p. 615)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.3.2. Turbulence Modeling in Swirling Flows

If you are modeling turbulent flow with a significant amount of swirl (for example, cyclone flows, swirling jets), you should consider using one of ANSYS Fluent's advanced turbulence models: the RNG k- ε model, realizable k- ε model, or Reynolds stress model. The appropriate choice depends on the strength of the swirl, which can be gauged by the swirl number. The swirl number is defined as the ratio of the axial flux of angular momentum to the axial flux of axial momentum:

$$S=\frac{\int rw\vec{v}\cdot d\vec{A}}{\overline{R}\int u\vec{v}\cdot d\vec{A}} \tag{8.15}$$

where $\overline{R}$ is the hydraulic radius.

For flows with weak to moderate swirl ($S<0.5$), both the RNG k- ε model and the realizable k- ε model yield appreciable improvements over the standard k- ε model. See RNG k-ε Model and Realizable k-ε Model Swirl Modification (p. 801) for details about these models.

For highly swirling flows ($S>0.5$), the Reynolds stress model (RSM) is strongly recommended. The effects of strong turbulence anisotropy can be modeled rigorously only by the second-moment closure adopted in the RSM. See Reynolds Stress Model (RSM) Steps in Using a Turbulence Model (p. 764) for details about this model.

For swirling flows encountered in devices such as cyclone separators and swirl combustors, near-wall turbulence modeling is quite often a secondary issue at most. The fidelity of the predictions in these cases is mainly determined by the accuracy of the turbulence model in the core region. However, in cases where walls actively participate in the generation of swirl (that is, where the secondary flows and vortical flows are generated by pressure gradients), non-equilibrium wall functions can often improve the predictions since they use a law of the wall for mean velocity sensitized to pressure gradients. See Near-Wall Treatments for Wall-Bounded Turbulent Flows in the Theory Guide for additional details about near-wall treatments for turbulence.

8.3.3. Mesh Setup for Swirling and Rotating Flows

8.3.3.1. Coordinate System Restrictions

Recall that for an axisymmetric problem, the axis of rotation must be the x axis and the mesh must lie on or above the $y=0$ line.

8.3.3.2. Mesh Sensitivity in Swirling and Rotating Flows

In addition to the setup constraint described above, you should be aware of the need for sufficient resolution in your mesh when solving flows that include swirl or rotation. Typically, rotating boundary layers may be very thin, and your ANSYS Fluent model will require a very fine mesh near a rotating wall. In addition, swirling flows will often involve steep gradients in the circumferential velocity (for example, near the centerline of a free-vortex type flow), and therefore require a fine mesh for accurate resolution.

8.3.4. Modeling Axisymmetric Flows with Swirl or Rotation

As discussed in Overview of Swirling and Rotating Flows (p. 588), you can solve a 2D axisymmetric problem that includes the prediction of the circumferential or swirl velocity. The assumption of

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

axisymmetry implies that there are no circumferential gradients in the flow, but that there may be nonzero circumferential velocities. Examples of axisymmetric flows involving swirl or rotation are depicted in Figure 8.10: Rotating Flow in a Cavity (p. 590) and Figure 8.11: Swirling Flow in a Gas Burner (p. 590).

Figure 8.10: Rotating Flow in a Cavity

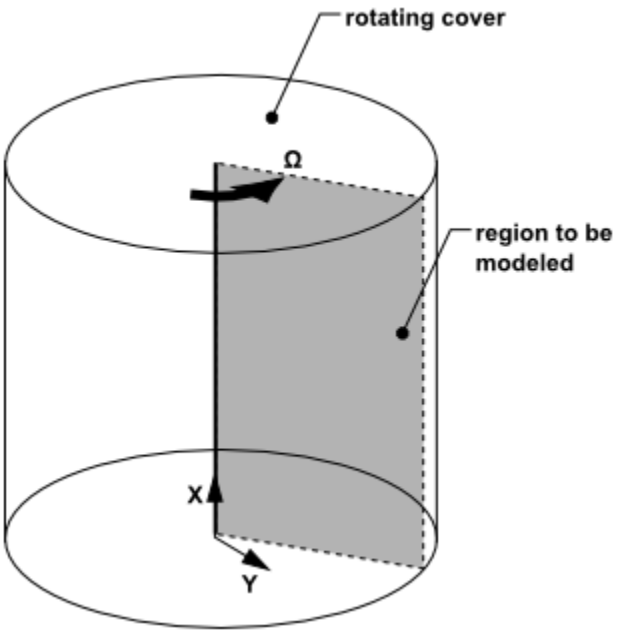
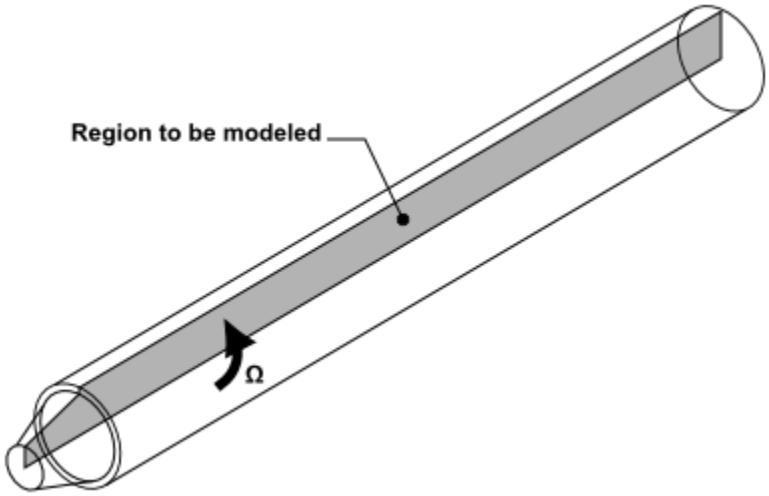


Figure 8.11: Swirling Flow in a Gas Burner



8.3.4.1. Problem Setup for Axisymmetric Swirling Flows

For axisymmetric problems, you must perform the following steps during the problem setup procedure. (Only those steps relevant specifically to the setup of axisymmetric swirl/rotation are listed here. You must set up the rest of the problem as usual.)

1. Enable the solution of the momentum equation in the circumferential direction by turning on the **Axisymmetric Swirl** option for **Space** in the **General** task page.

   Setup → **General** → **Axisymmetric Swirl**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Define the rotational or swirling component of velocity, $r\Omega$, at inlets or walls.

Setup → Boundary Conditions

Important

Remember to use the axis boundary type for the axis of rotation.

The procedures for input of rotational velocities at inlets and at walls are described in detail in Defining the Velocity (p. 318) and Velocity Conditions for Moving Walls (p. 359).

8.3.4.2. Solution Strategies for Axisymmetric Swirling Flows

The difficulties associated with solving swirling and rotating flows are a result of the high degree of coupling between the momentum equations, which is introduced when the influence of the rotational terms is large. A high level of rotation introduces a large radial pressure gradient that drives the flow in the axial and radial directions. This, in turn, determines the distribution of the swirl or rotation in the field. This coupling may lead to instabilities in the solution process, and you may require special solution techniques in order to obtain a converged solution. Solution techniques that may be beneficial in swirling or rotating flow calculations include the following:

- (Pressure-based segregated solver only) Use the PRESTO! scheme (enabled in the **Pressure** list for **Spatial Discretization** in the Solution Methods Task Page (p. 2504)), which is well-suited for the steep pressure gradients involved in swirling flows.
- Ensure that the mesh is sufficiently refined to resolve large gradients in pressure and swirl velocity.
- (Pressure-based solver only) Change the under-relaxation parameters on the velocities, perhaps to 0.3–0.5 for the radial and axial velocities and 0.8–1.0 for swirl.
- (Pressure-based solver only) Use a sequential or step-by-step solution procedure, in which some equations are temporarily left inactive (see below).
- If necessary, start the calculations using a low rotational speed or inlet swirl velocity, increasing the rotation or swirl gradually in order to reach the final desired operating condition (see below).

See Using the Solver (p. 1551) for details on the procedures used to make these changes to the solution parameters. More details on the step-by-step procedure and on the gradual increase of the rotational speed are provided below.

8.3.4.2.1. Step-By-Step Solution Procedures for Axisymmetric Swirling Flows

Often, flows with a high degree of swirl or rotation will be easier to solve if you use the following step-by-step solution procedure, in which only selected equations are left active in each step. This approach allows you to establish the field of angular momentum, then leave it fixed while you update the velocity field, and then finally to couple the two fields by solving all equations simultaneously.

Important

Since the density-based solvers solve all the flow equations simultaneously, the following procedure applies only to the pressure-based solver.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In this procedure, you will use the **Equations...** button in the Solution Controls Task Page (p. 2508) to turn individual transport equations on and off between calculations.

1. If your problem involves inflow/outflow, begin by solving the flow without rotation or swirl effects. That is, enable the **Axisymmetric** option instead of the **Axisymmetric Swirl** option in the General Task Page (p. 2160), and do not set any rotating boundary conditions. The resulting flow-field data can be used as a starting guess for the full problem.

2. Enable the **Axisymmetric Swirl** option and set all rotating/swirling boundary conditions.

3. Begin the prediction of the rotating/swirling flow by solving only the momentum equation describing the circumferential velocity. This is the **Swirl Velocity** listed in the **Equations** list in the Equations Dialog Box (p. 2511). Let the rotation "diffuse" throughout the flow field, based on your boundary condition inputs. In a turbulent flow simulation, you may also want to leave the turbulence equations active during this step. This step will establish the field of rotation throughout the domain.

4. Turn off the momentum equations describing the circumferential motion (**Swirl Velocity**). Leaving the velocity in the circumferential direction fixed, solve the momentum and continuity (pressure) equations (**Flow** in the **Equations** list in the Equations Dialog Box (p. 2511)) in the other coordinate directions. This step will establish the axial and radial flows that are a result of the rotation in the field. Again, if your problem involves turbulent flow, you should leave the turbulence equations active during this calculation.

5. Turn on all of the equations simultaneously to obtain a fully coupled solution. Note the under-relaxation controls suggested above.

In addition to the steps above, you may want to simplify your calculation by solving isothermal flow before adding heat transfer or by solving laminar flow before adding a turbulence model. These two methods can be used for any of the solvers (that is, pressure-based or density-based).

8.3.4.2.2. Improving Solution Stability by Gradually Increasing the Rotational or Swirl Speed

Because the rotation or swirl defined by the boundary conditions can lead to large complex forces in the flow, your ANSYS Fluent calculations will be less stable as the speed of rotation or degree of swirl increases. Hence, one of the most effective controls you can apply to the solution is to solve your rotating flow problem starting with a low rotational speed or swirl velocity and then slowly increase the magnitude up to the desired level. The procedure for accomplishing this is as follows:

1. Set up the problem using a low rotational speed or swirl velocity in your inputs for boundary conditions. The rotation or swirl in this first attempt might be selected as 10% of the actual operating conditions.

2. Solve the problem at these conditions, perhaps using the step-by-step solution strategy outlined above.

3. Save this initial solution data.

4. Modify your inputs (boundary conditions). Increase the speed of rotation, perhaps doubling it.

5. Restart the calculation using the solution data saved in step 3 as the initial solution for the new calculation. Save the new data.

6. Continue to increment the speed of rotation, following steps 4 and 5, until you reach the desired operating condition.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.3.4.2.2.1. Postprocessing for Axisymmetric Swirling Flows

Reporting of results for axisymmetric swirling flows is the same as for other flows. The following additional variables are available for postprocessing when axisymmetric swirl is active:

- **Swirl Velocity** (in the **Velocity...** category)
- **Swirl-Wall Shear Stress** (in the **Wall Fluxes...** category)

8.4. Compressible Flows

Compressibility effects are encountered in gas flows at high velocity and/or in which there are large pressure variations. When the flow velocity approaches or exceeds the speed of sound of the gas or when the pressure change in the system ($\Delta p / p$) is large, the variation of the gas density with pressure has a significant impact on the flow velocity, pressure, and temperature. Compressible flows create a unique set of flow physics for which you must be aware of the special input requirements and solution techniques described in this section. Figure 8.12: Transonic Flow in a Converging-Diverging Nozzle (p. 593) and Figure 8.13: Mach 0.675 Flow Over a Bump in a 2D Channel (p. 594) show examples of compressible flows computed using ANSYS Fluent.

Figure 8.12: Transonic Flow in a Converging-Diverging Nozzle

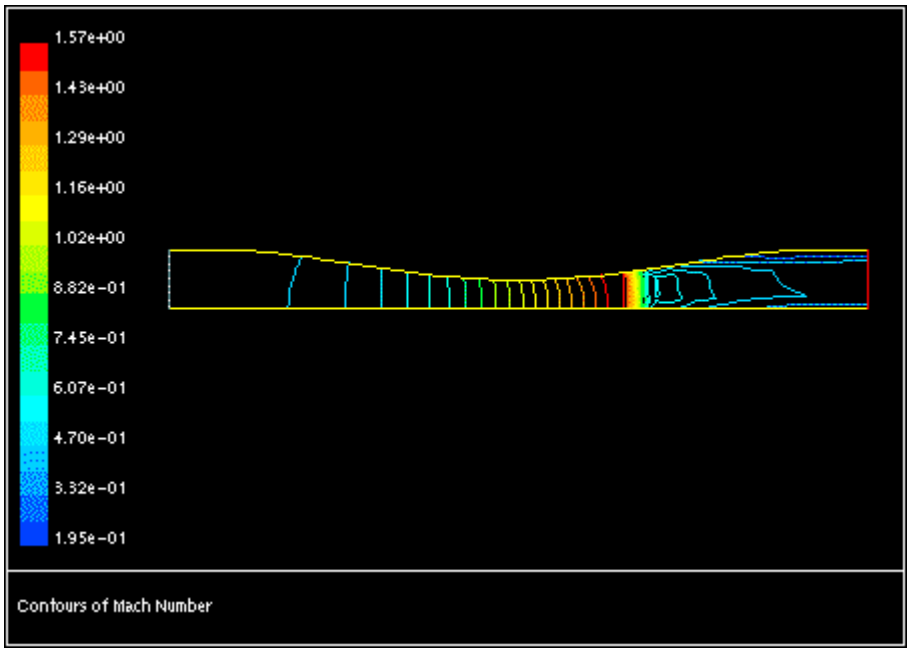
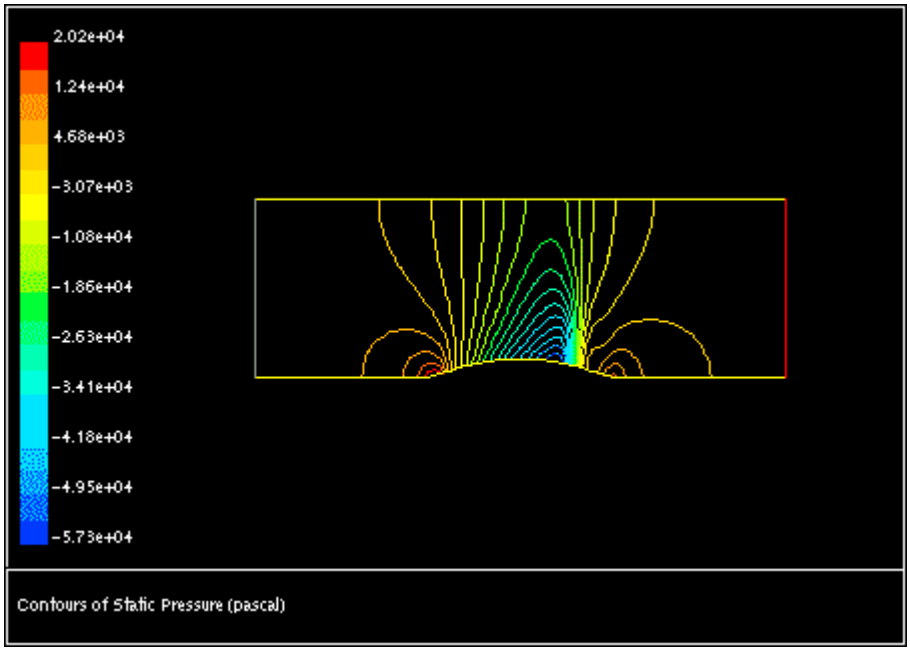


Figure 8.13: Mach 0.675 Flow Over a Bump in a 2D Channel



Information about compressible flows is provided in the following subsections:

8.4.1. When to Use the Compressible Flow Model
8.4.2. Physics of Compressible Flows
8.4.3. Modeling Inputs for Compressible Flows
8.4.4. Floating Operating Pressure
8.4.5. Solution Strategies for Compressible Flows
8.4.6. Reporting of Results for Compressible Flows

For more information about the theoretical background of compressible flows, see Compressible Flows in the Theory Guide.

8.4.1. When to Use the Compressible Flow Model

Compressible flows can be characterized by the value of the Mach number:

$$M \equiv u / c \tag{8.16}$$

Here, c is the speed of sound in the gas:

$$c = \sqrt{\gamma RT} \tag{8.17}$$

and γ is the ratio of specific heats $\left(c_p / c_v\right)$.

When the Mach number is less than 1.0, the flow is termed subsonic. At Mach numbers much less than 1.0 (M 0.1 or so), compressibility effects are negligible and the variation of the gas density with pressure can safely be ignored in your flow modeling. As the Mach number approaches 1.0 (which is referred to as the transonic flow regime), compressibility effects become important. When the Mach number exceeds 1.0, the flow is termed supersonic, and may contain shocks and expansion fans which can impact the flow pattern significantly. ANSYS Fluent provides a wide range of compressible flow modeling capabilities for subsonic, transonic, and supersonic flows.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.4.2. Physics of Compressible Flows

Compressible flows are typically characterized by the total pressure p_0 and total temperature T_0 of the flow. For an ideal gas, these quantities can be related to the static pressure and temperature by the following:

$$\frac{p_0}{p} = exp\left(\frac{\int_T^{T_0} \frac{C_p}{T} dT}{R}\right) \tag{8.18}$$

For constant C_p, Equation 8.18 (p. 595) reduces to

$$\begin{aligned} \frac{p_0}{p} &= \left(1+\frac{\gamma-1}{2}M^2\right)^{\gamma/(\gamma-1)} \\ \frac{T_0}{T} &= 1+\frac{\gamma-1}{2}M^2 \end{aligned} \tag{8.19}$$

These relationships describe the variation of the static pressure and temperature in the flow as the velocity (Mach number) changes under isentropic conditions. For example, given a pressure ratio from inlet to exit (total to static), Equation 8.19 (p. 595) can be used to estimate the exit Mach number that would exist in a one-dimensional isentropic flow. For air, Equation 8.19 (p. 595) p/p_0, of 0.5283. This choked flow condition will be established at the point of minimum flow area (for example, in the throat of a nozzle). In the subsequent area expansion the flow may either accelerate to a supersonic flow in which the pressure will continue to drop, or return to subsonic flow conditions, decelerating with a pressure rise. If a supersonic flow is exposed to an imposed pressure increase, a shock will occur, with a sudden pressure rise and deceleration accomplished across the shock.

8.4.2.1. Basic Equations for Compressible Flows

Compressible flows are described by the standard continuity and momentum equations solved by ANSYS Fluent, and you do not need to enable any special physical models (other than the compressible treatment of density as detailed below). The energy equation solved by ANSYS Fluent correctly incorporates the coupling between the flow velocity and the static temperature, and should be activated whenever you are solving a compressible flow. In addition, if you are using the pressure-based solver, you should enable the viscous dissipation terms in Equation 5.1 in the Theory Guide, which become important in high-Mach-number flows.

8.4.2.2. The Compressible Form of the Gas Law

For compressible flows, the ideal gas law is written in the following form:

$$\rho = \frac{p_{op}+p}{\frac{R}{M_w}T} \tag{8.20}$$

where p_{op} is the operating pressure defined in the Operating Conditions Dialog Box (p. 2382), p is the local static pressure relative to the operating pressure, R is the universal gas constant, and M_w is the molecular weight. The temperature, T, will be computed from the energy equation.

Some compressible flow problems involve fluids that do not behave as ideal gases. For example, flow under very high-pressure conditions cannot typically be modeled accurately using the ideal-gas assumption. Therefore, the real gas model described in Real Gas Models (p. 532) should be used instead.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.4.3. Modeling Inputs for Compressible Flows

To set up a compressible flow in ANSYS Fluent, you must follow the steps listed below. (Only those steps relevant specifically to the setup of compressible flows are listed here. You must set up the rest of the problem as usual.)

1. Set the **Operating Pressure** in the Operating Conditions Dialog Box (p. 2382).

   Setup → **Boundary Conditions** → **Operating Conditions...**

   (You can think of p_{op} as the absolute static pressure at a point in the flow where you will define the gauge pressure p to be zero. See Operating Pressure (p. 530) for guidelines on setting the operating pressure. For time-dependent compressible flows, you may want to specify a floating operating pressure instead of a constant operating pressure. See Floating Operating Pressure (p. 597) for details.)

2. Enable the solution of the energy equation.

   Setup → **Models** → **Energy** **On**

3. (Pressure-based solver only) If you are modeling turbulent flow, enable the optional viscous dissipation terms in the energy equation by turning on **Viscous Heating** in the Viscous Model Dialog Box (p. 2176). Note that these terms can be important in high-speed flows.

   Setup → **Models** → **Viscous** → **Edit...**

   This step is not necessary if you are using one of the density-based solvers, because the density-based solvers always include the viscous dissipation terms in the energy equation.

4. Set the following items in the Create/Edit Materials Dialog Box (p. 2302):

   Setup → **Materials** → **Create/Edit...**

   a. Select **ideal-gas** in the drop-down list next to **Density**.

   b. Define all relevant properties (specific heat, molecular weight, thermal conductivity, and so on).

5. Set cell zone conditions and boundary conditions (using the Boundary Conditions Task Page (p. 2389) and Cell Zone Conditions Task Page (p. 2367)), being sure to choose a well-posed cell zone or boundary condition combination that is appropriate for the flow regime. See below for details. Recall that all inputs for pressure (either total pressure or static pressure) must be relative to the operating pressure, and the temperature inputs at inlets should be total (stagnation) temperatures, *not* static temperatures.

   Setup → **Cell Zone Conditions**

   Setup → **Boundary Conditions**

These inputs should ensure a well-posed compressible flow problem. You will also want to consider special solution parameter settings, as noted in Solution Strategies for Compressible Flows (p. 599), before beginning the flow calculation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.4.3.1. Boundary Conditions for Compressible Flows

Well-posed inlet and exit boundary conditions for compressible flow are listed below:

- For flow inlets:
  - Pressure inlet: Inlet total temperature and total pressure and, for supersonic inlets, static pressure
  - Mass-flow inlet: Inlet mass flow and total temperature
- For flow exits:
  - Pressure outlet: Exit static pressure (ignored if flow is supersonic at the exit. All the information travels downstream in a supersonic region, hence the pressure at the outlet can be computed by directly extrapolating from the adjacent cell center [43] (p. 2889). Therefore, it is not meaningful to use the exit static pressure prescribed in the boundary conditions task page, and the exit static pressure is ignored).
  - Mass-flow outlet: Outlet mass flow

It is important to note that your boundary condition inputs for pressure (either total pressure or static pressure) must be in terms of gauge pressure — that is, pressure relative to the operating pressure defined in the Operating Conditions Dialog Box (p. 2382), as described above.

All temperature inputs at inlets should be total (stagnation) temperatures, *not* static temperatures.

8.4.4. Floating Operating Pressure

ANSYS Fluent provides a "floating operating pressure" option to handle time-dependent compressible flows with a gradual increase in the absolute pressure in the domain. This option is desirable for slow subsonic flows with static pressure build-up, since it efficiently accounts for the slow changing of absolute pressure without using acoustic waves as the transport mechanism for the pressure build-up.

Examples of typical applications include the following:

- combustion or heating of a gas in a closed domain
- pumping of a gas into a closed domain

8.4.4.1. Limitations

The floating operating pressure option should *not* be used for transonic or incompressible flows. In addition, it cannot be used if your model includes any pressure inlet, pressure outlet, exhaust fan, inlet vent, intake fan, outlet vent, or pressure far field boundaries.

8.4.4.2. Theory

The floating operating pressure option allows ANSYS Fluent to calculate the pressure rise (or drop) from the integral mass balance, separately from the solution of the pressure correction equation. When this option is activated, the absolute pressure at each iteration can be expressed as

$$p_{abs} = p_{op,float} + p \tag{8.21}$$

where p is the pressure relative to the reference location, which in this case is in the cell with the minimum pressure value. Therefore the reference location itself is floating.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$p_{op,float}$ is referred to as the floating operating pressure, and is defined as

$$p_{op,float} = p_{op}^{0} + \Delta p_{op} \tag{8.22}$$

where p_{op}^{0} is the initial operating pressure and Δp_{op} is the pressure rise.

Including the pressure rise Δp_{op} in the floating operating pressure $p_{op,float}$, rather than in the pressure p, helps to prevent roundoff error. If the pressure rise were included in p, the calculation of the pressure gradient for the momentum equation would give an inexact balance due to precision limits for 32-bit real numbers.

8.4.4.3. Enabling Floating Operating Pressure

When time dependence is active, you can turn on the **Floating Operating Pressure** option in the Operating Conditions Dialog Box (p. 2382).

Setup → **Boundary Conditions** → **Operating Conditions...**

(Note that the inputs for **Reference Pressure Location** will disappear when you enable **Floating Operating Pressure**, since these inputs are no longer relevant.)

Important

The floating operating pressure option should *not* be used for transonic flows or for incompressible flows. It is meaningful only for slow subsonic flows of ideal gases, when the characteristic time scale is much larger than the sonic time scale.

8.4.4.4. Setting the Initial Value for the Floating Operating Pressure

When the floating operating pressure option is enabled, you must specify a value for the **Initial Operating Pressure** in the Solution Initialization Task Page (p. 2520).

Solution → **Initialization**

This initial value is stored in the case file with all your other initial values.

8.4.4.5. Storage and Reporting of the Floating Operating Pressure

The current value of the floating operating pressure is stored in the data file. If you visit the Operating Conditions Dialog Box (p. 2382) after a number of time steps have been performed, the current value of the **Operating Pressure** will be displayed.

Note that the floating operating pressure will automatically be reset to the initial operating pressure if you reset the data (that is, start over at the first iteration of the first time step).

8.4.4.6. Monitoring Absolute Pressure

You can monitor the absolute pressure during the calculation using the Surface Report Definition Dialog Box (p. 2807). You can also generate graphical plots or alphanumeric reports of absolute pressure when your solution is complete. The **Absolute Pressure** variable is contained in the **Pressure...** category of

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the variable selection drop-down list that appears in postprocessing dialog boxes. See Field Function Definitions (p. 1931) for its definition.

8.4.5. Solution Strategies for Compressible Flows

The difficulties associated with solving compressible flows are a result of the high degree of coupling between the flow velocity, density, pressure, and energy. This coupling may lead to instabilities in the solution process and, therefore, may require special solution techniques in order to obtain a converged solution. In addition, the presence of shocks (discontinuities) in the flow introduces an additional stability problem during the calculation. Solution techniques that may be beneficial in compressible flow calculations include the following:

- (Pressure-based solver only) Initialize the flow to be near stagnation (that is velocity small but not zero, pressure to inlet total pressure, temperature to inlet total temperature). Turn off the energy equation for the first 50 iterations. Leave the energy under-relaxation at 1. Set the pressure under-relaxation to 0.4, and the momentum under-relaxation to 0.3. After the solution stabilizes and the energy equation has been turned on, increase the pressure under-relaxation to 0.7.

- Set reasonable limits for the temperature and pressure (in the Solution Limits Dialog Box (p. 2511)) to avoid solution divergence, especially at the start of the calculation. If ANSYS Fluent prints messages about temperature or pressure being limited as the solution nears convergence, the high or low computed values may be physical, and you must change the limits to allow these values.

- If required, begin the calculations using a reduced pressure ratio at the boundaries, increasing the pressure ratio gradually in order to reach the final desired operating condition. If the Mach number is low, you can also consider starting the compressible flow calculation from an incompressible flow solution (although the incompressible flow solution can in some cases be a rather poor initial guess for the compressible calculation).

- In some cases, computing an inviscid solution as a starting point may be helpful.

See Using the Solver (p. 1551) for details on the procedures used to make these changes to the solution parameters.

8.4.6. Reporting of Results for Compressible Flows

You can display the results of your compressible flow calculations in the same manner that you would use for an incompressible flow. The variables listed below are of particular interest when you model compressible flow:

- **Total Temperature**

- **Total Pressure**

- **Mach Number**

These variables are contained in the variable selection drop-down list that appears in postprocessing dialog boxes. **Total Temperature** is in the **Temperature...** category, **Total Pressure** is in the **Pressure...** category, and **Mach Number** is in the **Velocity...** category. See Field Function Definitions (p. 1931) for their definitions.

8.5. Inviscid Flows

Inviscid flow analysis neglect the effect of viscosity on the flow and are appropriate for high-Reynolds-number applications where inertial forces tend to dominate viscous forces. One example for which an

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

inviscid flow calculation is appropriate is an aerodynamic analysis of some high-speed projectile. In a case like this, the pressure forces on the body will dominate the viscous forces. Hence, an inviscid analysis will give you a quick estimate of the primary forces acting on the body. After the body shape has been modified to maximize the lift forces and minimize the drag forces, you can perform a viscous analysis to include the effects of the fluid viscosity and turbulent viscosity on the lift and drag forces.

Another area where inviscid flow analysis are routinely used is to provide a good initial solution for problems involving complicated flow physics and/or complicated flow geometry. In a case like this, the viscous forces are important, but in the early stages of the calculation the viscous terms in the momentum equations will be ignored. Once the calculation has been started and the residuals are decreasing, you can turn on the viscous terms (by enabling laminar or turbulent flow) and continue the solution to convergence. For some very complicated flows, this is the only way to get the calculation started.

Information about inviscid flows is provided in the following subsections:

8.5.1. Setting Up an Inviscid Flow Model
8.5.2. Solution Strategies for Inviscid Flows
8.5.3. Postprocessing for Inviscid Flows

For more information about the theoretical background of inviscid flows, see Inviscid Flows in the Theory Guide.

8.5.1. Setting Up an Inviscid Flow Model

For inviscid flow problems, you must perform the following steps during the problem setup procedure. (Only those steps relevant specifically to the setup of inviscid flow are listed here. You must set up the rest of the problem as usual.)

1. Enable the calculation of inviscid flow by selecting **Inviscid** in the Viscous Model Dialog Box (p. 2176).

   Setup → **Models** → **Viscous** → **Model** → **Inviscid**

2. Set boundary conditions and flow properties.

   Setup → **Boundary Conditions**

   Note

   Walls are assumed to be slip surfaces (the velocity is not equal to zero, unlike viscous flows) and therefore have a tangential velocity computed based on the solution of the governing equations.

   Setup → **Materials**

3. Solve the problem and examine the results.

8.5.2. Solution Strategies for Inviscid Flows

Since inviscid flow problems will usually involve high-speed flow, you may have to reduce the under-relaxation factors for momentum (if you are using the pressure-based solver) or reduce the Courant number (if you are using the density-based solver), in order to get the solution started. Once the flow

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

is started and the residuals are monotonically decreasing, you can start increasing the under-relaxation factors or Courant number back up to the default values.

Modifications to the under-relaxation factors and the Courant number can be made in the Solution Controls Task Page (p. 2508).

Solution → Controls

The solution strategies for compressible flows apply also to inviscid flows. See Solution Strategies for Compressible Flows (p. 599) for details.

8.5.3. Postprocessing for Inviscid Flows

If you are interested in the lift and drag forces acting on your model, you can use the Force Reports Dialog Box (p. 2614) to compute them.

Results → Reports → Forces Edit...

See Forces on Boundaries (p. 1916) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 9: Modeling Flows with Moving Reference Frames

This chapter provides details about the moving reference frame capabilities in ANSYS Fluent.

The information in this chapter is divided into the following sections:

9.1. Introduction
9.2. Flow in Single Moving Reference Frames (SRF)
9.3. Flow in Multiple Moving Reference Frames

9.1. Introduction

ANSYS Fluent solves the equations of fluid flow and heat transfer, by default, in a stationary (or inertial) reference frame. However, there are many problems where it is advantageous to solve the equations in a moving (or non-inertial) reference frame. Such problems typically involve moving parts (such as rotating blades, impellers, and similar types of moving surfaces), and it is the flow around these moving parts that is of interest. In most cases, the moving parts render the problem unsteady when viewed from the stationary frame. With a moving reference frame, however, the flow around the moving part can (with certain restrictions) be modeled as a steady-state problem with respect to the moving frame.

ANSYS Fluent's moving reference frame modeling capability allows you to model problems involving moving parts by allowing you to enable moving reference frames in selected cell zones. When a moving reference frame is activated, the equations of motion are modified to incorporate the additional acceleration terms that occur due to the transformation from the stationary to the moving reference frame. By solving these equations in a steady-state manner, the flow around the moving parts can be modeled.

For many problems, it may be possible to refer the entire computational domain to a single moving reference frame. This is known as the single reference frame (or SRF) approach. The use of the SRF approach is possible; provided the geometry meets certain requirements. For more complex geometries, it may not be possible to use a single reference frame. In such cases, you must break up the problem into multiple cell zones, with well-defined interfaces between the zones. The manner in which the interfaces are treated leads to two approximate, steady-state modeling methods for this class of problem: the multiple reference frame (or MRF) approach, and the mixing plane approach. These approaches will be discussed in The Multiple Reference Frame Model (p. 613) and The Mixing Plane Model (p. 615). If unsteady interaction between the stationary and moving parts is important, you can employ the sliding mesh approach to capture the transient behavior of the flow. The sliding meshing model will be discussed in Modeling Flows Using Sliding and Dynamic Meshes (p. 627).

The principal reason for employing a moving reference frame is to render a problem that is unsteady in the stationary (inertial) frame, steady with respect to the moving frame. For a steadily moving frame (for example, the frame speed is constant), it is possible to transform the equations of fluid motion to the moving frame such that steady-state solutions are possible. By default, ANSYS Fluent permits the activation of a moving reference frame with a steady speed. If the speed is not constant, the transformed equations will contain additional terms (see Relative Velocity Formulation in the Theory Guide). It should also be noted that you can run an unsteady simulation in a moving reference frame with constant speed. This would be necessary if you wanted to simulate, for example, vortex shedding from a rotating fan blade. The unsteadiness in this case is due to a natural fluid instability (vortex generation) rather than induced from interaction with a stationary component.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For more information about the equations for moving reference frames, see Equations for a Moving Reference Frame in the Theory Guide.

Figure 9.1: Single Component (Blower Wheel Blade Passage)

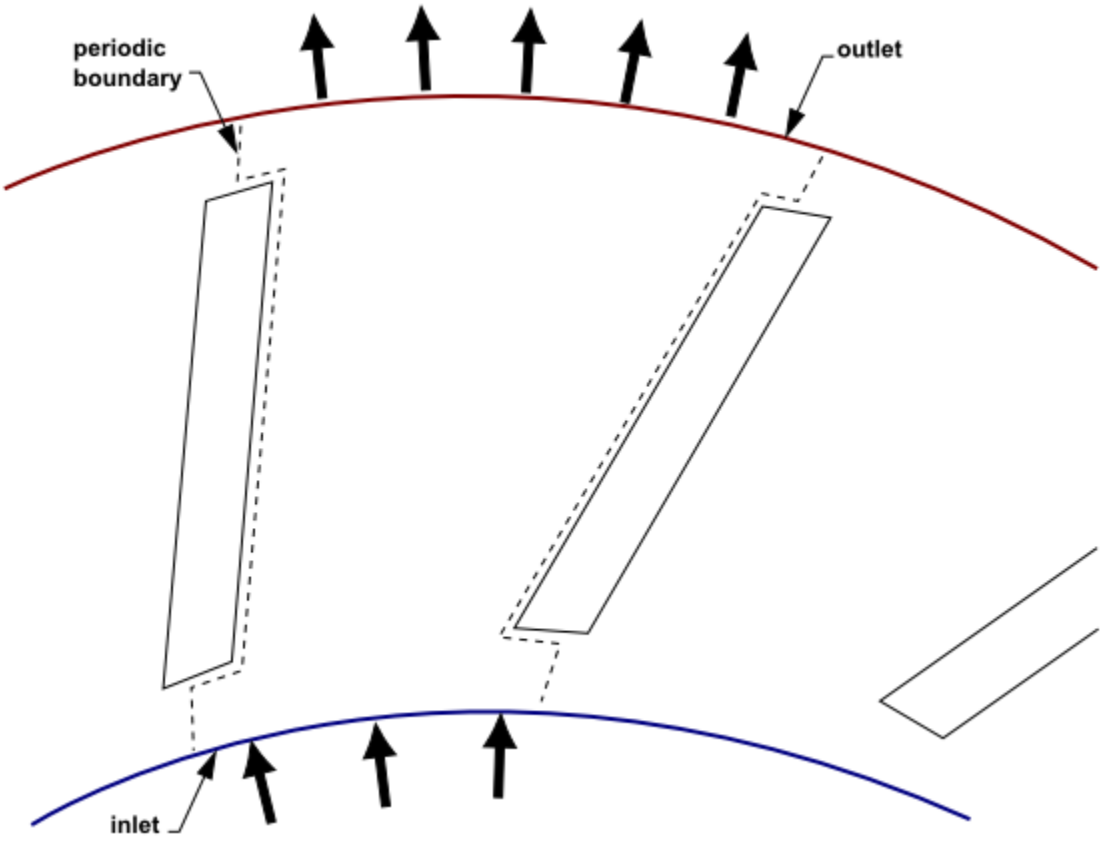
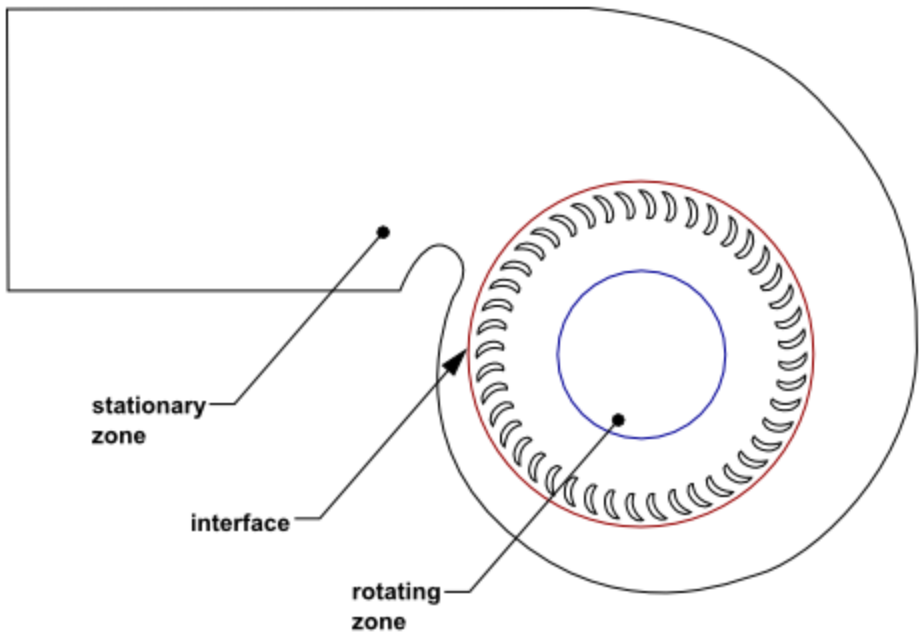


Figure 9.2: Multiple Component (Blower Wheel and Casing)



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

9.2. Flow in Single Moving Reference Frames (SRF)

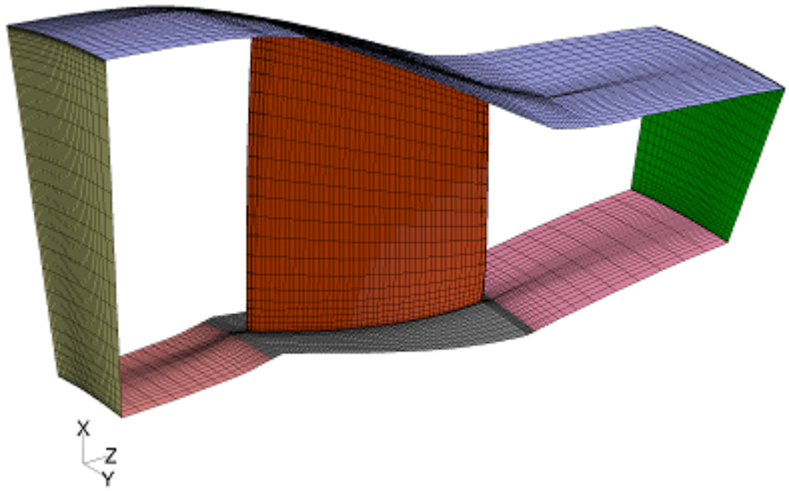
Many problems permit the entire computational domain to be referred to a single moving reference frame (hence the name SRF modeling). In such cases, the equations given in Equations for a Moving Reference Frame are solved in all fluid cell zones. Steady-state solutions are possible in SRF models provided suitable boundary conditions are prescribed. In particular, wall boundaries must adhere to the following requirements:

- Any walls that are moving with the reference frame can assume any shape. An example would be the blade surfaces associated with a pump impeller. The no slip condition is defined in the relative frame such that the relative velocity is zero on the moving walls.
- For a rotating problem, you can define walls that are non-moving with respect to the stationary coordinate system, but these walls must be surfaces of revolution about the axis of rotation. Here the no slip condition is defined such that the absolute velocity is zero on the walls. An example of this type of boundary would be a cylindrical wind tunnel wall that surrounds a rotating propeller.

Rotationally periodic boundaries may also be used, but the surface must be periodic about the axis of rotation. As an example, it is very common to model flow through a blade row of a turbomachine by assuming the flow to be rotationally periodic and using a periodic domain about a single blade. This permits good resolution of the flow around the blade without the expense of modeling all blades in the blade row (see Figure 9.3: Single Blade Model with Rotationally Periodic Boundaries (p. 605)).

Flow boundary conditions in ANSYS Fluent (inlets and outlets) can, in most cases, be prescribed in either the stationary or moving frames. For example, for a velocity inlet, one can specify either the relative velocity or absolute velocity, depending on which is more convenient. For additional information on these and other boundary conditions, see Setting Up a Single Moving Reference Frame Problem (p. 606) and Cell Zone and Boundary Conditions (p. 237).

Figure 9.3: Single Blade Model with Rotationally Periodic Boundaries



9.2.1. Mesh Setup for a Single Moving Reference Frame

It is important to remember the following coordinate-system constraints when you are setting up a problem involving a moving reference frame for a rotating problem:

- For 2D problems, the axis of rotation must be parallel to the z axis.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For 2D axisymmetric problems, the axis of rotation must be the x axis.
- For 3D geometries, you should generate the mesh with a specific origin and rotational axis in mind for the rotating cell zone. Usually it is convenient to use the origin of the global coordinate system (0,0,0) for the frame origin, and either the x, y, or z axis for the rotational axis; however, ANSYS Fluent can accommodate an arbitrary origin and rotational axis.

With 3D rotating problems, it is also important to note that if you want to include walls that have zero velocity in the stationary frame, these walls must be a surface of revolution with respect to the axis of rotation. If the stationary walls are not surfaces of revolution, you must encapsulate the rotating parts with interface boundaries, thereby breaking your model up into multiple zones, and use either the MRF or mixing plane models for steady-state solutions (see The Multiple Reference Frame Model (p. 613) and The Mixing Plane Model (p. 615)), or the sliding mesh model for unsteady interaction (see Modeling Flows Using Sliding and Dynamic Meshes (p. 627)).

9.2.2. Setting Up a Single Moving Reference Frame Problem

To model a problem involving a single moving reference frame, follow the steps outlined below.

1. Select the **Velocity Formulation** to be used when solving: either **Relative** or **Absolute**. (See Choosing the Relative or Absolute Velocity Formulation (p. 609) for details.)

   Setup → **General**

   (Note that this step is irrelevant if you are using one of the density-based solvers; these solvers always use an absolute velocity formulation.)

2. For each cell zone in the domain, specify the translational velocity of the reference frame and/or the angular velocity (ω) of the reference frame and the axis about which it rotates.

   Setup → **Cell Zone Conditions**

   a. In the **Fluid** or **Solid** dialog box, specify the **Rotation-Axis Origin** and **Rotation-Axis Direction** for the frame motion to define the axis of rotation.

   b. Also in the **Fluid** (Figure 9.4: The Fluid Dialog Box Displaying Frame Motion Inputs (p. 608)) or **Solid** dialog box, enable the **Frame Motion** option and then set the **Speed** under **Rotational Velocity** and/or the **X**, **Y**, and **Z** components of the **Translational Velocity** in the expanded portion of the dialog box under the **Reference Frame** tab. Note that the speed can be specified as a constant value or a transient profile. The transient profile may be in a file format, as described in Defining Transient Cell Zone and Boundary Conditions (p. 448), or a UDF macro, described in `DEFINE_TRANSIENT_PROFILE`. Specifying the individual velocities as either a profile or a UDF allows you to specify a specific input of the frame motion individually. However, you can also specify the frame motion inputs via a single user-defined function that uses the UDF macro `DEFINE_ZONE_MOTION`. This may prove to be quite convenient if you are modeling a more complicated motion of the moving reference frame, where the hooking of many different user-defined functions or profiles can be cumbersome.

   Note

   If you decide to hook a UDF, you will no longer have access to the rotation axis origin and direction, or the velocities.

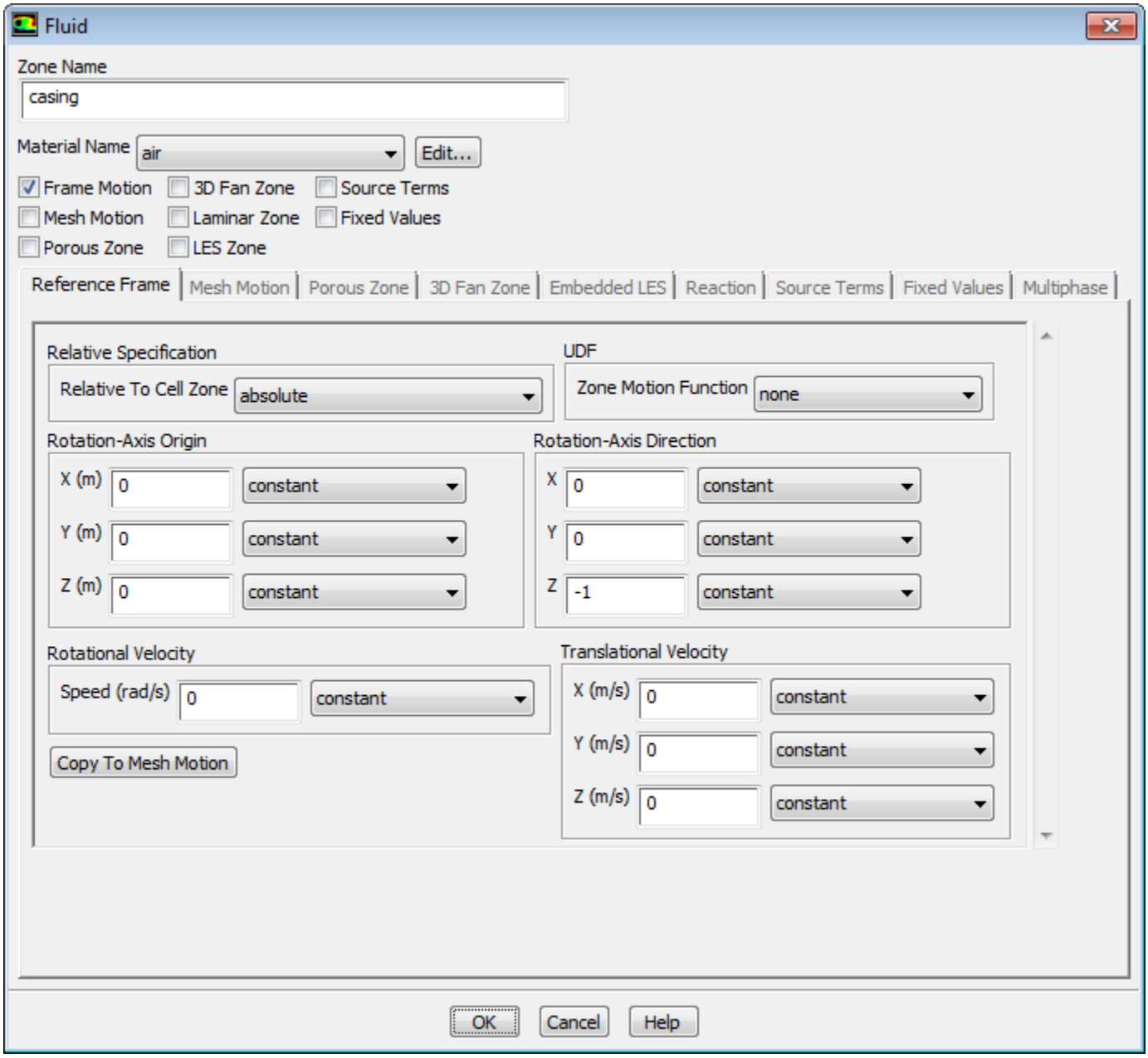
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Details about these inputs are presented in Inputs for Fluid Zones (p. 253) and in Inputs for Solid Zones (p. 259). Details about the zone motion UDF can be found in `DEFINE_ZONE_MOTION` in the Fluent Customization Manual.

c. If you need to switch between the moving reference frame and moving mesh models, simply click the **Copy To Mesh Motion** for zones with a moving frame of reference and **Copy to Frame Motion** for zones with moving meshes to transfer motion variables, such as the axes, frame origin, and velocity components between the two models. The variables used for the origin, axis, and velocity components, as well as for the UDF `DEFINE_ZONE_MOTION` will be copied. This is particularly useful if you are doing a steady-state MRF simulation to obtain an initial solution for a transient Moving Mesh simulation in a turbomachine.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 9.4: The Fluid Dialog Box Displaying Frame Motion Inputs



Important

For solid zones, you only need to enable the **Frame Motion** option if you intend to include the convective terms in the energy equation for the solid (Equation 5.11 in the Theory Guide). Normally, this is not required if you want to do a conjugate heat transfer problem where the solid and fluid zones are moving together.

3. Define the velocity boundary conditions at walls. You can choose to define either an absolute velocity or a velocity relative to the moving reference frame (that is, relative to the velocity of the adjacent cell zone specified in step 2).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the wall is moving at the speed of the moving frame (and hence stationary in the moving frame), it is convenient to specify a relative angular velocity of zero. Likewise, a wall that is stationary in the non-moving frame of reference should be given a velocity of zero in the absolute reference frame. Specifying the wall velocities in this manner obviates the need to modify these inputs later if a change is made in the velocity of the fluid zone.

Details about these inputs are presented in Velocity Conditions for Moving Walls (p. 359).

4. Define the boundary conditions at the inlets, as described in Boundary Conditions (p. 301). For velocity inlets, you can choose to define either absolute velocities or velocities relative to the motion of the adjacent cell zone (specified in step 2). Likewise, the total pressure and flow direction can be prescribed in absolute or relative frames for pressure inlets.

   Details about these inputs are presented in Defining the Flow Direction (p. 311) and Defining the Velocity (p. 318).

9.2.2.1. Choosing the Relative or Absolute Velocity Formulation

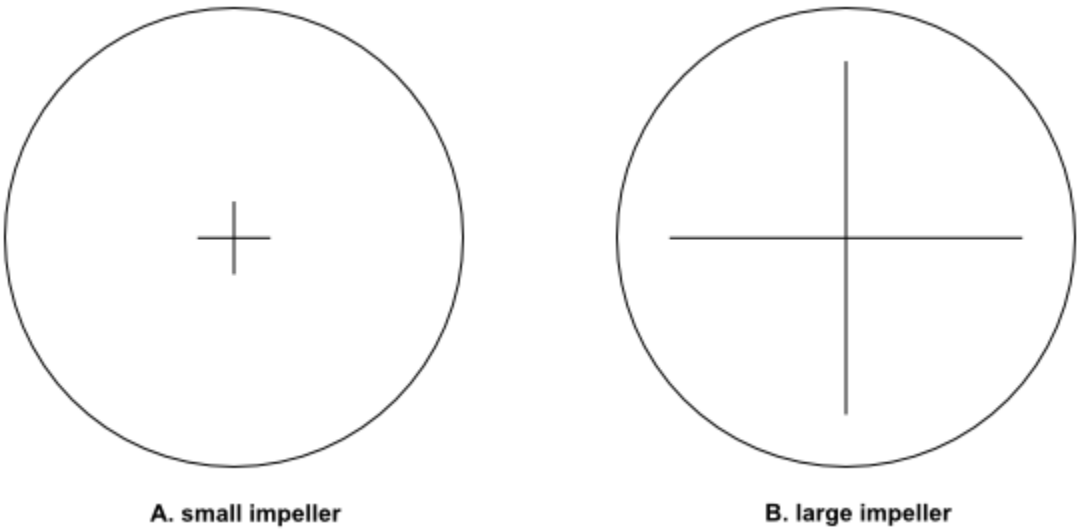
It is recommended that you use the velocity formulation that will result in most of the flow domain having the smallest velocities in that frame, thereby reducing the numerical diffusion in the solution and leading to a more accurate solution.

The absolute velocity formulation is preferred in applications where the flow in most of the domain is not moving (for example, a fan in a large room). The relative velocity formulation is appropriate when most of the fluid in the domain is moving, as in the case of a large impeller in a mixing tank.

9.2.2.1.1. Example

A problem with stationary outer walls and a rotating impeller can be solved in a single reference frame. The example is illustrated in Figure 9.5: Geometry with the Rotating Impeller (p. 609).

Figure 9.5: Geometry with the Rotating Impeller



In case A, it is expected that only the flow near the impeller would be rotating and that much of the flow away from the impeller would have a low velocity magnitude in the absolute frame. Therefore, solving using the absolute velocity formulation is recommended. In case B, most of the flow is expected

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

to be rotating with a velocity close to that of the impeller. Hence, the relative velocity formulation is appropriate.

In a situation between case A and case B, either of the formulations may be used.

Important

- If the velocity formulation is switched during the solution process, ANSYS Fluent will not transform the current solution to the other frame, which can lead to large jumps in residuals. If changing the frame is necessary, it is recommended that you first reinitialize, and then solve.
- When one of the density-based solution algorithms is used, the absolute formulation is always used; the relative velocity formulation is not available in the density-based solvers.

For velocity inlets, pressure inlets, mass flow inlets, and walls, you may specify velocity in either the absolute or the relative frame, regardless of whether the absolute or relative velocity formulation is used in the computation.

For pressure outlets, the specified static pressure is independent of frame. However, when there is backflow at a pressure outlet, the specified static pressure is used as the total pressure. For calculations using the absolute velocity formulation, the specified static pressure is used as the total pressure in the absolute frame; for the relative velocity formulation, the specified static pressure is assumed to be the total pressure in the relative frame. As for the flow direction, ANSYS Fluent assumes the absolute velocity to be normal to the pressure outlet for the absolute velocity formulation; for the relative velocity formulation, it is the relative velocity that is assumed to be normal to the pressure outlet.

9.2.3. Solution Strategies for a Single Moving Reference Frame

The difficulties associated with solving flows in moving reference frames are similar to those discussed in Solution Strategies for Axisymmetric Swirling Flows (p. 591). The primary issue you must confront is the high degree of coupling between the momentum equations when the influence of the rotational terms is large. A high degree of rotation introduces a large radial pressure gradient that drives the flow in the axial and radial directions, thereby setting up a distribution of the swirl or rotation in the field. This coupling may lead to instabilities in the solution process, and hence require special solution techniques to obtain a converged solution. Some techniques that may be beneficial include the following:

- (Pressure-based solver only) Consider switching the frame in which velocities are solved by changing the velocity formulation setting in the General Task Page (p. 2160). (See Choosing the Relative or Absolute Velocity Formulation (p. 609) for details.)
- (Pressure-based segregated solver only) Use the PRESTO! scheme (enabled in the Solution Methods Task Page (p. 2504)), which is well-suited for the steep pressure gradients involved in rotating flows.
- Ensure that the mesh is sufficiently refined to resolve large gradients in pressure and swirl velocity.
- (Pressure-based, segregated solver only) Reduce the under-relaxation factors for the velocities, perhaps to 0.3–0.5 or lower, if necessary.
- Begin the calculations using a low rotational speed, increasing the rotational speed gradually in order to reach the final desired operating condition.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See Using the Solver (p. 1551) for details on the procedures used to make these changes to the solution parameters.

9.2.3.1. Gradual Increase of the Rotational Speed to Improve Solution Stability

Because the rotation of the reference frame and the rotation defined via boundary conditions can lead to large complex forces in the flow, your ANSYS Fluent calculations may be less stable as the speed of rotation (and hence the magnitude of these forces) increases. One of the most effective controls you can exert on the solution is to start with a low rotational speed and then slowly increase the rotation up to the desired level. The procedure you use to accomplish this is as follows:

1. Set up the problem using a low rotational speed in your inputs for boundary conditions and for the angular velocity of the reference frame. The rotational speed in this first attempt might be selected as 10% of the actual operating condition.

2. Solve the problem at these conditions.

3. Save this initial solution data.

4. Modify your inputs (that is, boundary conditions and angular velocity of the reference frame). Increase the speed of rotation, perhaps doubling it.

5. Restart or continue the calculation using the solution data saved in Step 3 as the initial guess for the new calculation. Save the new data.

6. Continue to increment the rotational speed, following Steps 4 and 5, until you reach the desired operating condition.

9.2.4. Postprocessing for a Single Moving Reference Frame

When you solve a problem in a moving reference frame, you can plot or report both absolute and relative velocities. For all velocity parameters (for example, **Velocity Magnitude** and **Mach Number**), corresponding relative values will be available for postprocessing (for example, **Relative Velocity Magnitude** and **Relative Mach Number**). These variables are contained in the **Velocity...** category of the variable selection drop-down list that appears in postprocessing dialog boxes. Relative values are also available for postprocessing of total pressure, total temperature, and any other parameters that include a dynamic contribution dependent on the reference frame (for example, **Relative Total Pressure**, **Relative Total Temperature**, **Rothalpy**).

When plotting velocity vectors, you can choose to plot vectors in the absolute frame (the default), or you can select **Relative Velocity** in the **Vectors of** drop-down list in the Vectors Dialog Box (p. 2831) to plot vectors in the moving frame. If you plot relative velocity vectors, you might want to color the vectors by relative velocity magnitude (by choosing **Relative Velocity Magnitude** in the **Color by** list); by default they will be colored by absolute velocity magnitude. Figure 9.6: Absolute Velocity Vectors (p. 612) and Figure 9.7: Relative Velocity Vectors (p. 612) show absolute and relative velocity vectors in a moving domain with a stationary outer wall.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 9.6: Absolute Velocity Vectors

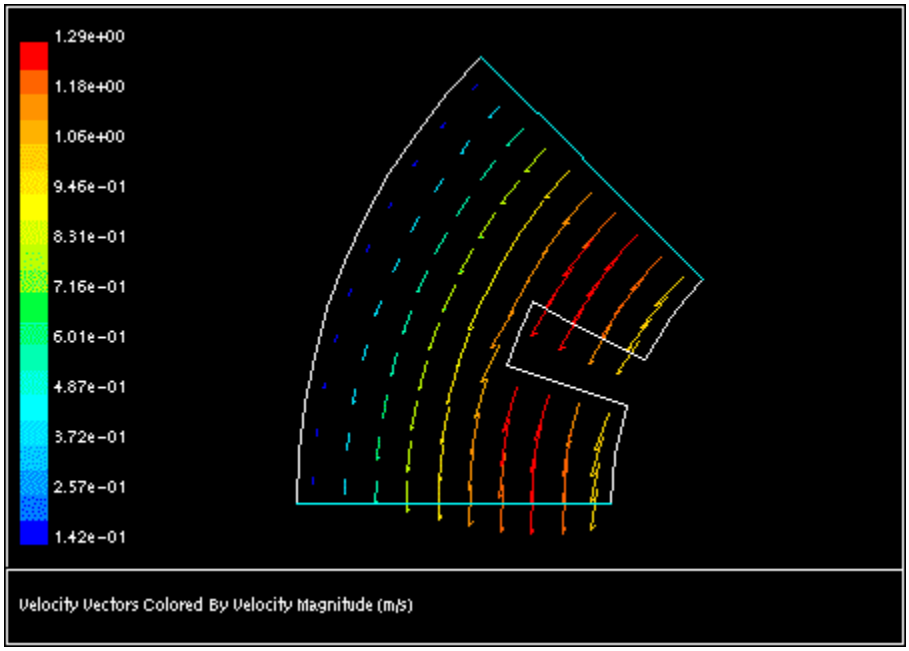
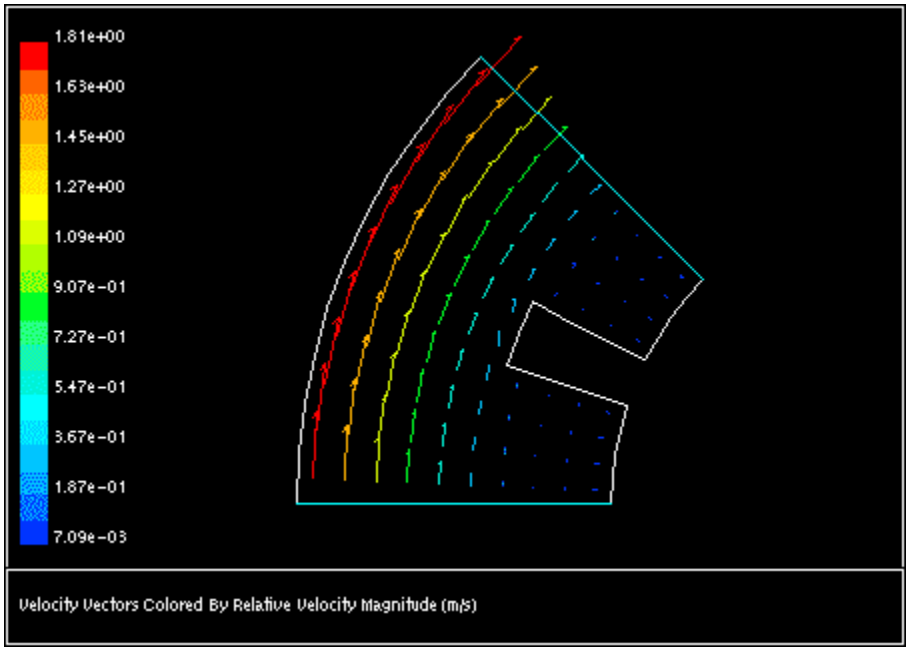


Figure 9.7: Relative Velocity Vectors



9.3. Flow in Multiple Moving Reference Frames

Many problems involve multiple moving parts or contain stationary surfaces which are not surfaces of revolution (and therefore cannot be used with the Single Reference Frame modeling approach). For these problems, you must break up the model into multiple fluid/solid cell zones, with interface boundaries separating the zones. Zones that contain the moving components can then be solved using the moving reference frame equations (Equations for a Moving Reference Frame in the Theory Guide), whereas stationary zones can be solved with the stationary frame equations. The manner in which the equations are treated at the interface lead to two approaches that are supported in ANSYS Fluent:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Multiple Moving Reference Frames
  - Multiple Reference Frame Model (MRF)
  - Mixing Plane Model (MPM)
- Sliding Mesh Model (SMM)

Both the MRF and mixing plane approaches are steady-state approximations, and differ primarily in the manner in which conditions at the interfaces are treated. These approaches will be discussed in the sections below. The sliding mesh model approach is, on the other hand, inherently unsteady due to the motion of the mesh with time. This approach is discussed in Modeling Flows Using Sliding and Dynamic Meshes (p. 627).

For additional information, see the following sections:
9.3.1. The Multiple Reference Frame Model
9.3.2. The Mixing Plane Model
9.3.3. Mesh Setup for a Multiple Moving Reference Frame
9.3.4. Setting Up a Multiple Moving Reference Frame Problem
9.3.5. Solution Strategies for MRF and Mixing Plane Problems
9.3.6. Postprocessing for MRF and Mixing Plane Problems

9.3.1. The Multiple Reference Frame Model

Additional information about the MRF model is presented in the following sections:
9.3.1.1. Overview
9.3.1.2. Limitations

9.3.1.1. Overview

The MRF model [63] (p. 2890) is, perhaps, the simplest of the two approaches for multiple zones. It is a steady-state approximation in which individual cell zones can be assigned different rotational and/or translational speeds. The flow in each moving cell zone is solved using the moving reference frame equations (see Introduction (p. 603)). If the zone is stationary (ω=0), the equations reduce to their stationary forms. At the interfaces between cell zones, a local reference frame transformation is performed to enable flow variables in one zone to be used to calculate fluxes at the boundary of the adjacent zone. For more information about the MRF interface formulation, see The MRF Interface Formulation in the Theory Guide.

It should be noted that the MRF approach does not account for the relative motion of a moving zone with respect to adjacent zones (which may be moving or stationary); the mesh remains fixed for the computation. This is analogous to freezing the motion of the moving part in a specific position and observing the instantaneous flowfield with the rotor in that position. Hence, the MRF is often referred to as the "frozen rotor approach."

While the MRF approach is clearly an approximation, it can provide a reasonable model of the flow for many applications. For example, the MRF model can be used for turbomachinery applications in which rotor-stator interaction is relatively weak, and the flow is relatively uncomplicated at the interface between the moving and stationary zones. In mixing tanks, for example, since the impeller-baffle interactions are relatively weak, large-scale transient effects are not present and the MRF model can be used.

Another potential use of the MRF model is to compute a flow field that can be used as an initial condition for a transient sliding mesh calculation. This eliminates the need for a startup calculation. The multiple reference frame model should not be used, however, if it is necessary to actually simulate the transients

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

that may occur in strong rotor-stator interactions, the sliding mesh model alone should be used (see Modeling Flows Using Sliding and Dynamic Meshes (p. 627).

For more information about and examples of multiple moving reference frames, see The Multiple Reference Frame Model in the Theory Guide.

9.3.1.2. Limitations

The following limitations exist when using the MRF approach:

- The interfaces separating a moving region from adjacent regions must be oriented such that the component of the frame velocity normal to the boundary is zero. This means that for a translationally moving frame, the moving zone's boundaries must be parallel to the translational velocity vector. For rotating problems, the interfaces must be surfaces of revolution about the axis of rotation defined for the fluid zone. For the example shown Figure 2.4: Geometry with One Rotating Impeller (in the Theory Guide), this requires the dashed boundary to be circular (not square or any other shape).

- Strictly speaking, the use of multiple reference frames is meaningful only for steady flow. However, ANSYS Fluent will allow you to solve an unsteady flow when multiple reference frames are being used. In this case, unsteady terms (as described in Temporal Discretization in the Theory Guide) are added to all the governing transport equations. You should carefully consider whether this will yield meaningful results for your application, because, for unsteady flows, a sliding mesh calculation will generally yield more meaningful results than an MRF calculation.

- By default, pathlines and particle trajectories drawn by ANSYS Fluent use the velocity relative to the reference frame motion of the cell zone. However, for particle trajectories, you can change the reference frame by either selecting **Track in Absolute Frame** in the **Numerics** tab of the **Discrete Phase Model** dialog box or using the text command `define/models/dpm/options/track-in-absolute-frame`.

  Note

  Only the default behavior (tracking relative to the moving reference frame) ensures that pathlines and massless particles follow the flow in a physically reasonable way; that is, pathlines or massless particles will not hit a wall surface with any strong wall-normal velocity component. Consistently with this, when a pathline or particle trajectory crosses a boundary between zones with different reference frame motion specifications, you may observe a sharp change in particle velocity as a result of the coordinate transformation.

  The particle injection velocities (specified in the Set Injection Properties Dialog Box (p. 2796)) are interpreted in the same way as tracking, that is, either relative to the local frame of reference's motion specification or in absolute coordinates.

- You cannot accurately model axisymmetric swirl in the presence of multiple reference frames using the relative velocity formulation. This is because the current implementation does not apply the transformation used in Equation 2.16 (in the Theory Guide) to the swirl velocity derivatives. For this situation, the absolute velocity formulation should be used.

- Translational and rotational velocities are assumed to be constant (time varying ω, v_t are not allowed).

- The relative velocity formulation cannot be used in combination with the MRF and mixture models. (For details, see Mixture Model Theory in the Theory Guide). For such cases, use the absolute velocity formulation instead.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You must not have a single interface between reference frames where part of the interface is made up of a coupled two-sided wall, while another part is not coupled (that is, the normal interface treatment). In such cases, you must break the interface up into two interfaces: one that is a coupled interface, and the other that is a standard fluid-fluid interface. See Using a Non-Conformal Mesh in ANSYS Fluent (p. 164) for the steps involved in setting up a coupled interface.

Important

You can switch from the MRF model to the sliding mesh model for a more robust and accurate solution, by using the `mesh/modify-zones/mrf-to-sliding-mesh` text command. See Using Sliding Meshes (p. 633) for details on how to make this change in the fluid's boundary conditions. Currently, this switch is not possible when running in parallel or for non-conformal interfaces.

9.3.2. The Mixing Plane Model

The mixing plane model in ANSYS Fluent provides an alternative to the multiple reference frame and sliding mesh models for simulating flow through domains with one or more regions in relative motion.

Additional information about the mixing plane model is presented in the following sections:
9.3.2.1. Overview
9.3.2.2. Limitations

9.3.2.1. Overview

As discussed in The Multiple Reference Frame Model (p. 613), the MRF model is applicable when the flow at the interface between adjacent moving/stationary zones is nearly uniform ("mixed out"). If the flow at this interface is not uniform, the MRF model may not provide a physically meaningful solution. The sliding mesh model (see Modeling Flows Using Sliding and Dynamic Meshes (p. 627)) may be appropriate for such cases, but in many situations it is not practical to employ a sliding mesh. For example, in a multistage turbomachine, if the number of blades is different for each blade row, a large number of blade passages is required in order to maintain circumferential periodicity. Moreover, sliding mesh calculations are necessarily unsteady, and therefore require significantly more computation to achieve a final, time-periodic solution. For situations where using the sliding mesh model is not feasible, the mixing plane model can be a cost-effective alternative.

In the mixing plane approach, each fluid zone is treated as a steady-state problem. Flow-field data from adjacent zones are passed as boundary conditions that are spatially averaged or "mixed" at the mixing plane interface. This mixing removes any unsteadiness that would arise due to circumferential variations in the passage-to-passage flow field (for example, wakes, shock waves, separated flow), therefore yielding a steady-state result. Despite the simplifications inherent in the mixing plane model, the resulting solutions can provide reasonable approximations of the time-averaged flow field.

9.3.2.2. Limitations

Note the following limitations of the mixing plane model:

- The LES turbulence model cannot be used with the mixing plane model.
- The models for species transport and combustion cannot be used with the mixing plane model.
- The VOF multiphase model cannot be used with the mixing plane model.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The discrete phase model cannot be used with the mixing plane model for coupled flows. Non-coupled computations can be done, but you should note that the particles leave the domain of the mixing plane.

You can find more information about the following topics in the Theory Guide:

Rotor and Stator Domains
The Mixing Plane Concept
Choosing an Averaging Method
Mixing Plane Algorithm of ANSYS Fluent
Mass Conservation
Swirl Conservation
Total Enthalpy Conservation

9.3.3. Mesh Setup for a Multiple Moving Reference Frame

Two mesh setup methods are available. Choose the method that is appropriate for your model, noting the restrictions in Limitations (p. 614).

- If the boundary between two zones that are in different reference frames is conformal (that is, the mesh node locations are identical at the boundary where the two zones meet), you can simply create the mesh as usual, with all cell zones contained in the same mesh file. A different cell zone should exist for each portion of the domain that is modeled in a different reference frame. Use an *interior* zone for the boundary between reference frames.

- If the boundary between two zones that are in different reference frames is non-conformal (that is, the mesh node locations are *not* identical at the boundary where the two zones meet), follow the non-conformal mesh setup procedure described in Using a Non-Conformal Mesh in ANSYS Fluent (p. 164).

9.3.4. Setting Up a Multiple Moving Reference Frame Problem

To learn more about setting up a multiple reference frame problem, see the following sections:
9.3.4.1. Setting Up Multiple Reference Frames
9.3.4.2. Setting Up the Mixing Plane Model

9.3.4.1. Setting Up Multiple Reference Frames

To model a problem involving multiple reference frames, perform the following:

Important

The mesh-setup constraints for a moving reference frame listed in Mesh Setup for a Single Moving Reference Frame (p. 605) apply to multiple reference frames as well.

---

1. Select the **Velocity Formulation** to be used in the General Task Page (p. 2160): either **Absolute** or **Relative**. (For details, see Choosing the Relative or Absolute Velocity Formulation (p. 609).)

   Setup → **General**

   (Note that this step is irrelevant if you are using one of the density-based solution algorithms; these algorithms always use an absolute velocity formulation.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. For each cell zone in the domain, specify its translational velocity and/or its angular velocity (ω) and the axis about which it rotates.

   Setup → Cell Zone Conditions

   a. If the zone is moving, or if you plan to specify cylindrical velocity or flow-direction components at inlets to the zone, you must define the axis of rotation of the frame of reference. In the **Fluid** or **Solid** dialog box, specify the **Rotation-Axis Origin** and **Rotation-Axis Direction** under the **Reference Frame** tab.

   b. In the **Fluid** (Figure 9.4: The Fluid Dialog Box Displaying Frame Motion Inputs (p. 608)) or **Solid** dialog box, enable the **Frame Motion** option. Set the **Speed** under **Rotational Velocity** and/or the **X**, **Y**, and **Z** components of the **Translational Velocity** in the expanded portion of the dialog box under the **Reference Frame** tab.

      Note that the speed can be specified as a constant value or a transient profile. The transient profile may be in a file format, as described in Defining Transient Cell Zone and Boundary Conditions (p. 448), or a UDF macro, described in `DEFINE_TRANSIENT_PROFILE`. Specifying the individual velocities as either a profile or a UDF allows you to specify a specific input of the frame motion individually. However, you can also specify the frame motion inputs via a single user-defined function that uses the UDF macro `DEFINE_ZONE_MOTION`. This may prove to be quite convenient if you are modeling a more complicated motion of the moving reference frame, where the hooking of many different user-defined functions or profiles can be cumbersome.

      Note

      If you decide to hook a UDF, then you will no longer have access to the rotation axis origin and direction, or the velocities.

      Details about these inputs are presented in Inputs for Fluid Zones (p. 253) and in Inputs for Solid Zones (p. 259). Details about the zone motion UDF can be found in `DEFINE_ZONE_MOTION` in the Fluent Customization Manual.

   c. To switch between the MRF and moving mesh models, click the **Copy To Mesh Motion** for zones with a moving frame of reference and **Copy to Frame Motion** for zones with moving meshes to transfer motion variables, such as the axes, frame origin, and velocity components between the two models.

      The variables used for the origin, axis, and velocity components, as well as for the UDF `DEFINE_ZONE_MOTION` will be copied. This is particularly useful if you are doing a steady-state MRF simulation to obtain an initial solution for a transient Moving Mesh simulation in a turbomachine.

3. Define the velocity boundary conditions at walls. You can choose to define either an absolute velocity or a velocity relative to the velocity of the adjacent cell zone specified in step 2.

   If the wall is moving at the speed of the moving frame (and hence stationary relative to the moving frame), it is convenient to specify a relative angular velocity of zero. Likewise, a wall that is stationary in the non-moving frame of reference should be given a velocity of zero in the absolute reference frame. Specifying the wall velocities in this manner obviates the need to modify these inputs later if a change is made in the rotational velocity of the fluid zone.

   An example for which you would specify a relative velocity is as follows: If an impeller is defined as **wall-3** and the fluid region within the impeller's radius is defined as **fluid-5**, you would need to specify the angular velocity and axis of rotation for **fluid-5** and then assign **wall-3** a relative velocity

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of 0. If you later wanted to model a different angular velocity for the impeller, you would need to change only the angular velocity of the fluid region; you would not need to modify the wall velocity conditions.

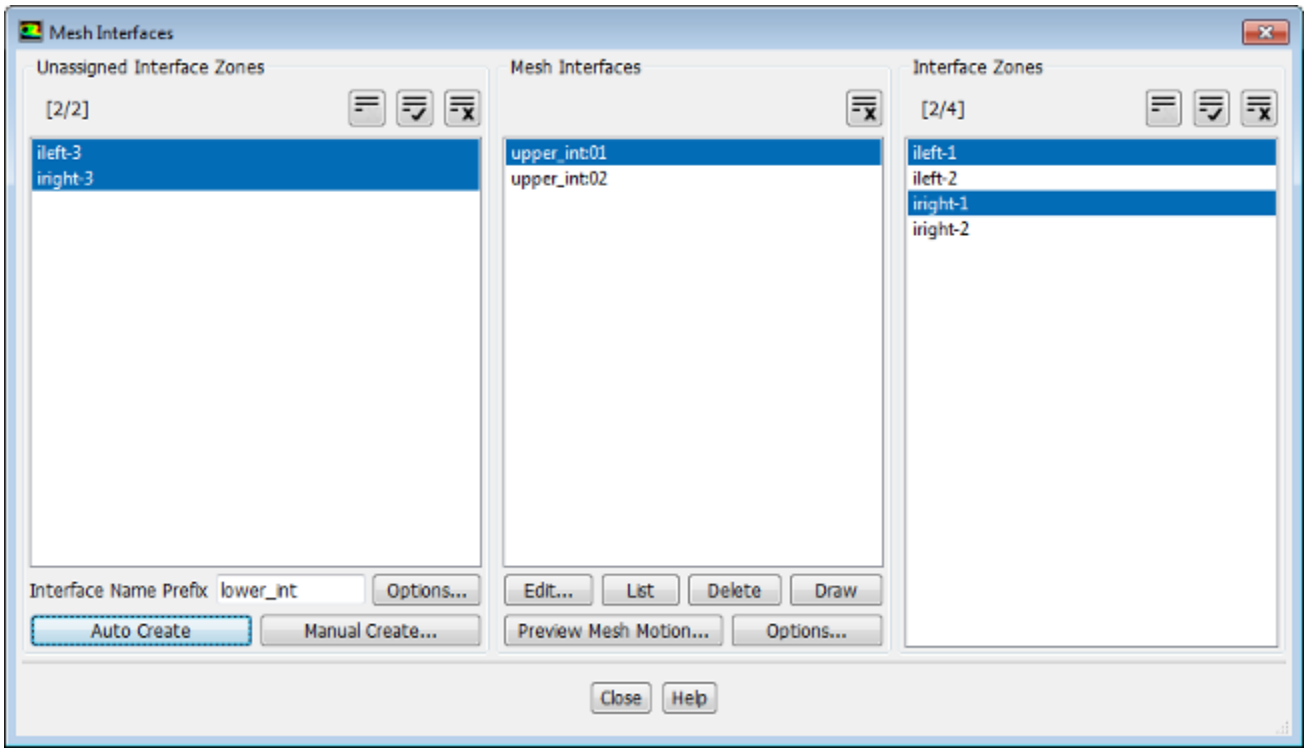
Details about these inputs are presented in Velocity Conditions for Moving Walls (p. 359).

4. Define the boundary conditions at the inlets, as described in Boundary Conditions (p. 301). For velocity inlets, you can choose to define either absolute velocities or velocities relative to the motion of the adjacent cell zone (specified in step 2). Likewise, the total pressure and flow direction can be prescribed in absolute or relative frames for pressure inlets.

   Details about these inputs are presented in Defining the Flow Direction (p. 311) and Defining the Velocity (p. 318).

5. Define the mesh interfaces using the Mesh Interfaces Dialog Box (p. 2732) (Figure 10.10: The Mesh Interfaces Dialog Box (p. 636)).

   Setup → Mesh Interfaces → New...



   To learn how to use the **Mesh Interfaces** dialog box, see Using a Non-Conformal Mesh in ANSYS Fluent (p. 164).

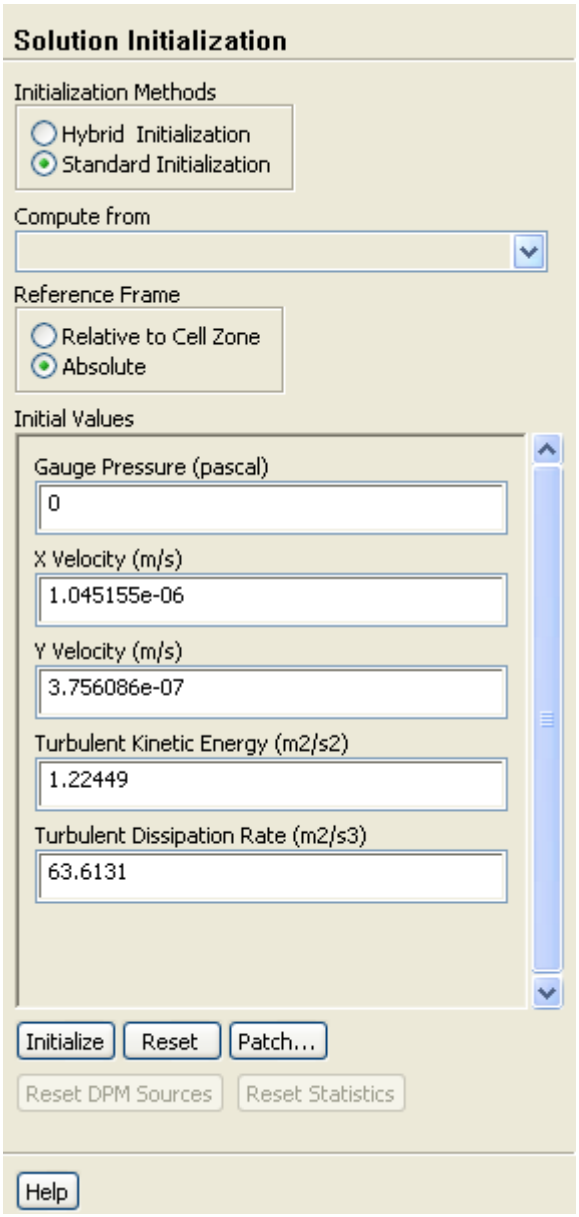
6. Initialize the solution using an absolute frame of reference (Figure 9.8: The Solution Initialization Task Page for Moving Reference Frames (p. 619)).

   Solution → Initialization

   Select the **Absolute** option under **Reference Frame**. If the **Relative to Cell Zone** option is selected, the initial flow field can contain discontinuities, which can cause convergence problems in the first few iterations.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 9.8: The Solution Initialization Task Page for Moving Reference Frames



9.3.4.2. Setting Up the Mixing Plane Model

The model inputs for mixing planes are presented in this section. Only those steps relevant specifically to the setup of a mixing plane problem are listed here. Note that the use of wall and periodic boundaries in a mixing plane model is consistent with their use when the model is not active.

1. Select the **Absolute** or **Relative Velocity Formulation** in the General Task Page (p. 2160), when the pressure-based solver is enabled.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → General

> **Important**
>
> When the density-based solver is enabled, only the **Absolute Velocity Formulation** can be used with the mixing plane model.

2. For each cell zone in the domain, specify its angular velocity (ω) and the axis about which it rotates.

   Setup → Cell Zone Conditions

   a. If the zone is rotating, or if you plan to specify cylindrical-velocity or flow-direction components at inlets to the zone, you must define the axis of rotation for the frame of reference. In the **Fluid** dialog box or Solid Dialog Box (p. 2379), specify the **Rotation-Axis Origin** and **Rotation-Axis Direction** under the **Reference Frame** tab.

   b. In the **Fluid** (Figure 9.4: The Fluid Dialog Box Displaying Frame Motion Inputs (p. 608)) or **Solid** dialog box, enable the **Frame Motion** option and then set the **Speed** under **Rotational Velocity** and/or the **X**, **Y**, and **Z** components of the **Translational Velocity** in the expanded portion of the dialog box under the **Reference Frame** tab.

   > **Important**
   >
   > It is important to define the axis of rotation for the cell zones on *both* sides of the mixing plane interface, including the stationary zone.

3. Define the velocity boundary conditions at walls, as described in step 3 of Setting Up Multiple Reference Frames (p. 616).

4. Define the boundary conditions at the inlets, as described in Boundary Conditions (p. 301). For velocity inlets, you can choose to define either absolute velocities or velocities relative to the motion of the adjacent cell zone (specified in step 2). For pressure inlets and mass flow inlets, the specification of the flow direction and total pressure will always be absolute, because the absolute velocity formulation is always used for mixing plane calculations. For a mass flow inlet, you do not need to specify the mass flow rate or mass flux. ANSYS Fluent will automatically select the **Mass Flux with Average Mass Flux** specification method and set the correct values when you create the mixing plane, as described in More About Mass Flux and Average Mass Flux (p. 325).

   Details about these inputs are presented in Defining the Flow Direction (p. 311), Defining the Velocity (p. 318), and Inputs at Mass-Flow Inlet Boundaries (p. 323).

   > **Important**
   >
   > Note that the outlet boundary zone at the mixing plane interface must be defined as a pressure outlet, and the inlet boundary zone at the mixing plane interface must be defined as either a velocity inlet (incompressible flow only), a pressure inlet, or a mass flow inlet. The overall inlet and exit boundary conditions can be any suitable combination permitted by the solver (for example, velocity inlet, pressure inlet, or mass flow inlet; pressure outlet). Remember, however, that if mass conservation across the mixing plane is important, you

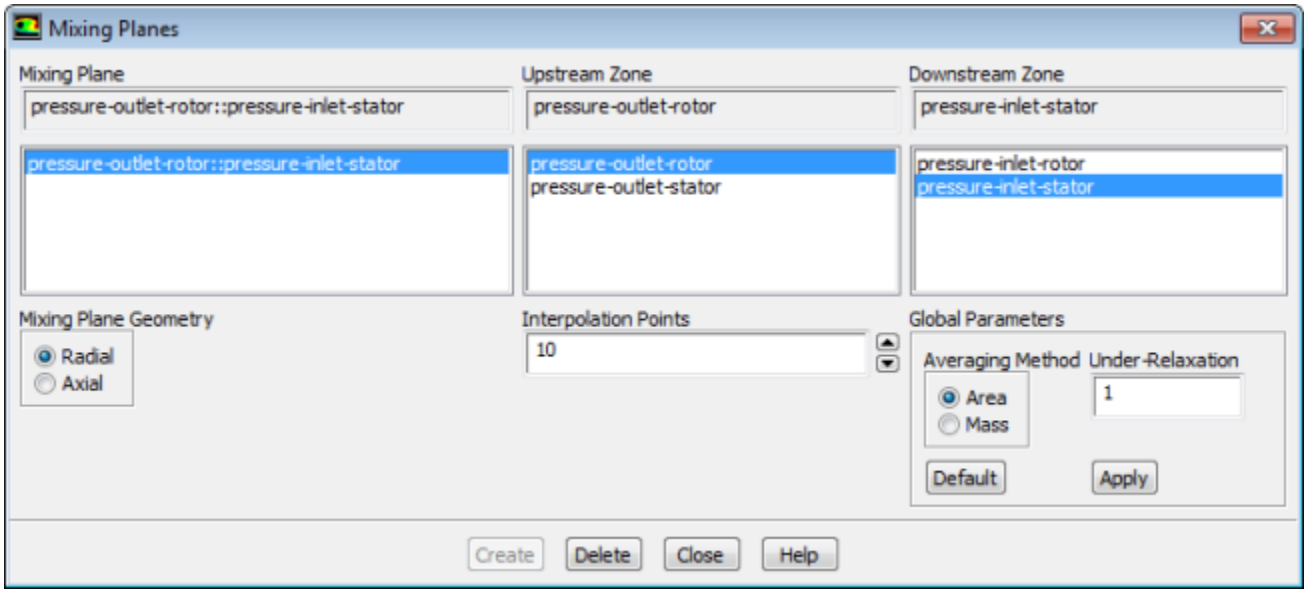
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

must use a mass flow inlet as the downstream boundary; strict mass conservation is *not* maintained across the mixing plane when you use a velocity inlet or pressure inlet.

5. Define the mixing planes in the Mixing Planes Dialog Box (p. 2744) (Figure 9.9: The Mixing Planes Dialog Box (p. 621)).

   Setting Up Domain → Mesh Models → Mixing Planes...

   Figure 9.9: The Mixing Planes Dialog Box



   a. Specify the two zones that comprise the mixing plane by selecting an upstream zone in the **Upstream Zone** list and a downstream zone in the **Downstream Zone** list. It is essential that the correct pairs be chosen from these lists (that is, that the boundary zones selected lie on the mixing plane interface). You can check this by displaying the mesh.

      Setup → General → Display...

   b. (3D only) Indicate the geometry of the mixing plane interface by choosing one of the options under **Mixing Plane Geometry**.

      A **Radial** geometry signifies that information at the mixing plane interface is to be circumferentially averaged into profiles that vary in the radial direction, for example, $p(r)$, $T(r)$. This is the case for axial-flow machines, for example.

      An **Axial** geometry signifies that circumferentially averaged profiles are to be constructed that vary in the axial direction, for example, $p(x)$, $T(x)$. This is the situation for a radial-flow device.

      Important

      Note that the radial direction is normal to the rotation axis for the fluid zone and the axial direction is parallel to the rotation axis.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. (3D only) Set the number of **Interpolation Points**. This is the number of radial or axial locations used in constructing the boundary profiles for circumferential averaging. You should choose a number that approximately corresponds to the resolution of the surface mesh in the radial or axial direction. Note that while you can use more points if you want, the resolution of the boundary profile will only be as fine as the resolution of the surface mesh itself.

   In 2D the flow data is averaged over the entire interface to create a profile consisting of a single data point. For this reason you do not need to set the number of **Interpolation Points** or select a **Mixing Plane Geometry** in 2D.

d. Set the **Global Parameters** for the mixing plane.

   i. Select the **Averaging Method**. The **Area** averaging method is the default method. For detailed information about each of the **Area**, **Mass**, or **Mixed-Out** options, see Choosing an Averaging Method in the Theory Guide.

   ii. Set the **Under-Relaxation** parameter. It is sometimes desirable to under-relax the changes in boundary values at mixing planes as these may change very rapidly during the early iterations of the solution and cause the calculation to diverge. The changes can be relaxed by specifying an under-relaxation less than 1. The new boundary profile values are then computed using

$$\phi_{new} = \phi_{old} + \alpha\left(\phi_{calculated} - \phi_{old}\right) \tag{9.1}$$

   where α is the under-relaxation factor. Once the flow field is established, the value of α can be increased.

   iii. Click **Apply** to set the **Global Parameters**. If the **Default** button is visible to the right of the **Apply** button, clicking the **Default** button will return **Global Parameters** back to their default values. The **Default** button will then change to be a **Reset** button. Clicking the **Reset** button will change the **Global Parameters** back to the values that were last applied.

e. Click **Create** to create a new mixing plane. ANSYS Fluent will name the mixing plane by combining the names of the zones selected as the **Upstream Zone** and **Downstream Zone** and enter the new mixing plane in the **Mixing Plane** list.

   If you create an incorrect mixing plane, you can select it in the **Mixing Plane** list and click the **Delete** button to delete it.

9.3.4.2.1. Modeling Options

There are two options available for use with the mixing plane model: a fixed pressure level for incompressible flows, and the swirl conservation described in Swirl Conservation in the Theory Guide.

9.3.4.2.1.1. Fixing the Pressure Level for an Incompressible Flow

For certain turbomachinery configurations, such as a torque converter, there is no fixed-pressure boundary when the mixing plane model is used. The mixing plane model is usually used to model the three interfaces that connect the components of the torque converter. In this configuration, the pressure is no longer fixed. As a result, the pressure may float unbounded, making it difficult to obtain a converged solution.

To resolve this problem, ANSYS Fluent offers an option for fixing the pressure level. When this option is enabled, ANSYS Fluent will adjust the gauge pressure field after each iteration by subtracting from

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

it the pressure value in the cell closest to the **Reference Pressure Location** in the Operating Conditions Dialog Box (p. 2382).

Important

This option is available only for incompressible flows calculated using the pressure-based solver.

To enable the fixed pressure option, use the `fix-pressure-level` text command:

`define` → `mixing-planes` → `set` → `fix-pressure-level`

9.3.4.2.1.2. Conserving Swirl Across the Mixing Plane

Conservation of swirl is important for applications such as torque converters (Swirl Conservation in the Theory Guide). If you want to enable swirl conservation across the mixing plane, you can use the commands in the `conserve-swirl` text menu:

`define` → `mixing-planes` → `set` → `conserve-swirl`

To turn on swirl conservation, use the `enable?` text command. Once the option is turned on, you can ask the solver to report information about the swirl conservation during the calculation. If you turn on `verbosity?`, ANSYS Fluent will report for every iteration the zone ID for the zone on which the swirl conservation is active, the upstream and downstream swirl integration per zone area, and the ratio of upstream to downstream swirl integration before and after the correction.

To obtain a report of the swirl integration at every pressure inlet, pressure outlet, velocity inlet, and mass flow inlet in the domain, use the `report-swirl-integration` command. You can use this information to determine the torque acting on each component of the turbomachinery according to Equation 2.22 (in the Theory Guide).

9.3.4.2.1.3. Conserving Total Enthalpy Across the Mixing Plane

One of the options available in the mixing plane model is to conserve total enthalpy across the mixing plane. This is a desirable feature because global parameters such as efficiency are directly related to the change in total enthalpy across a blade row or stage.

The procedure for ensuring conservation of total enthalpy simply involves adjusting the downstream total temperature profile such that the integrated total enthalpy matches the upstream integrated total enthalpy.

If you want to enable total enthalpy conservation, you can use the commands in the `conserve-total-enthalpy` text menu:

`define` → `mixing-planes` → `set` → `conserve-total-enthalpy`

To turn on total enthalpy conservation, use the `enable?` text command. Once the option is turned on, you can ask the solver to report information about the total enthalpy conservation during the calculation. If you turn on `verbosity?`, ANSYS Fluent will report at every iteration the zone ID for the zone on which the total enthalpy conservation is active, the upstream and downstream heat flux, and the ratio of upstream to downstream heat flux.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.3.5. Solution Strategies for MRF and Mixing Plane Problems

9.3.5.1. MRF Model

For multiple moving reference frames, follow the guidelines presented in Solution Strategies for a Single Moving Reference Frame (p. 610) for a single moving reference frame. Remember that with multiple zones, the possibility exists of interaction between moving and stationary components. This will manifest itself as poor or oscillatory convergence. In such cases, it is strongly recommend that the sliding mesh approach be used to compute the flowfield in order to resolve the unsteady interactions.

9.3.5.2. Mixing Plane Model

It should be emphasized that the mixing plane model is a reasonable approximation so long as there is no significant reverse flow in the vicinity of the mixing plane. If significant reverse flow occurs, the mixing plane will not be a satisfactory model of the actual flow. In a numerical simulation, reverse flow often occurs during the early stages of the computation even though the flow at convergence is not reversed. Therefore, it is helpful in these situations to first obtain a provisional solution using *fixed* conditions at the rotor-stator interface. The mixing plane model can then be enabled and the solution run to convergence.

If you are using the mass or mixed-out averaging method and you are experiencing convergence problems in the presence of severe reverse flow, initialize your solution using the default area-averaging method, then switch to mass or mixed-out averaging after the reverse flow dies out.

Under-relaxing the changes in the mixing plane boundary values can also help in troublesome situations. In many cases, setting the under-relaxation factor to a value less than 1 can be helpful. Once the flow field is established, you can gradually increase the under-relaxation factor.

9.3.6. Postprocessing for MRF and Mixing Plane Problems

When you solve a problem using the multiple reference frame or mixing plane model, you can plot or report both absolute and relative velocities. For all velocity parameters (for example, **Velocity Magnitude** and **Mach Number**), corresponding relative values will be available for postprocessing (for example, **Relative Velocity Magnitude** and **Relative Mach Number**). These variables are contained in the **Velocity...** category of the variable selection drop-down list that appears in postprocessing dialog boxes. Relative values are also available for postprocessing of total pressure, total temperature, and any other parameters that include a dynamic contribution dependent on the reference frame (for example, **Relative Total Pressure**, **Relative Total Temperature**).

Important

Relative velocities are relative to the translational/rotational velocity of the "reference zone" (specified in the **Reference Values** task page (see Reference Values Task Page (p. 2502))). The velocity of the reference zone is the velocity defined in the Fluid Dialog Box (p. 2369) for that zone.

When plotting velocity vectors, you can choose to plot vectors in the absolute frame (the default), or you can select **Relative Velocity** in the **Vectors of** drop-down list in the Vectors Dialog Box (p. 2831) to plot vectors relative to the translational/rotational velocity of the "reference zone" (specified in the Reference Values Task Page (p. 2502)). If you plot relative velocity vectors, you might want to color the vectors by relative velocity magnitude (by choosing **Relative Velocity Magnitude** in the **Color by** list); by default they will be colored by absolute velocity magnitude.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also generate a plot of circumferential averages in ANSYS Fluent. This allows you to find the average value of a quantity at several different radial or axial positions in your model. ANSYS Fluent computes the average of the quantity over a specified circumferential area, and then plots the average against the radial or axial coordinate. For more information on generating XY plots of circumferential averages, see XY Plots of Circumferential Averages (p. 1848).

For details about turbomachinery-specific postprocessing features see Turbomachinery Postprocessing (p. 1856).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 10: Modeling Flows Using Sliding and Dynamic Meshes

This chapter describes the setup and use of the sliding and dynamic mesh models in ANSYS Fluent. To learn more about the theory of sliding meshes in ANSYS Fluent, see Sliding Mesh Theory in the Theory Guide. For more information about the theory behind dynamic meshes in ANSYS Fluent, see Dynamic Mesh Theory in the Theory Guide.

Understanding and using the sliding and deforming mesh models is presented in the following sections:

10.1. Introduction
10.2. Sliding Mesh Examples
10.3. The Sliding Mesh Technique
10.4. Sliding Mesh Interface Shapes
10.5. Using Sliding Meshes
10.6. Using Dynamic Meshes

10.1. Introduction

The sliding mesh model allows you to set up a problem in which separate zones move relative to each other. The motion can be translational or rotational. The relative motion of stationary and moving components (for example, in a rotating machine) will give rise to transient interactions. Often, the transient solution that is sought in a sliding mesh simulation is time-periodic. That is, the transient solution repeats with a period related to the speeds of the moving domains.

The dynamic mesh model allows you to move the boundaries of a cell zone relative to other boundaries of the zone, and to adjust the mesh accordingly. The boundaries can move rigidly with respect to each other (that is, linear or rotational motion), and/or deform.

When deciding whether to use a sliding mesh versus a dynamic mesh, consider the following:

- Many problems could be solved by either approach.
- If the problem does not involve mesh deformation, the sliding mesh model is recommended, as it is simpler and more efficient.
- The dynamic mesh method must be used if the mesh is deforming, or if the mesh motion is a function of the solution (for example, the six degree of freedom solver).

Important

Sliding mesh and dynamic mesh are incompatible if they are combined within the same cell zone or adjacent cell zones. If you want to combine these two models within the same case file, they must be separated by a non-moving cell zone.

For examples of typical sliding mesh and dynamic mesh problems, see Introduction in the Theory Guide.

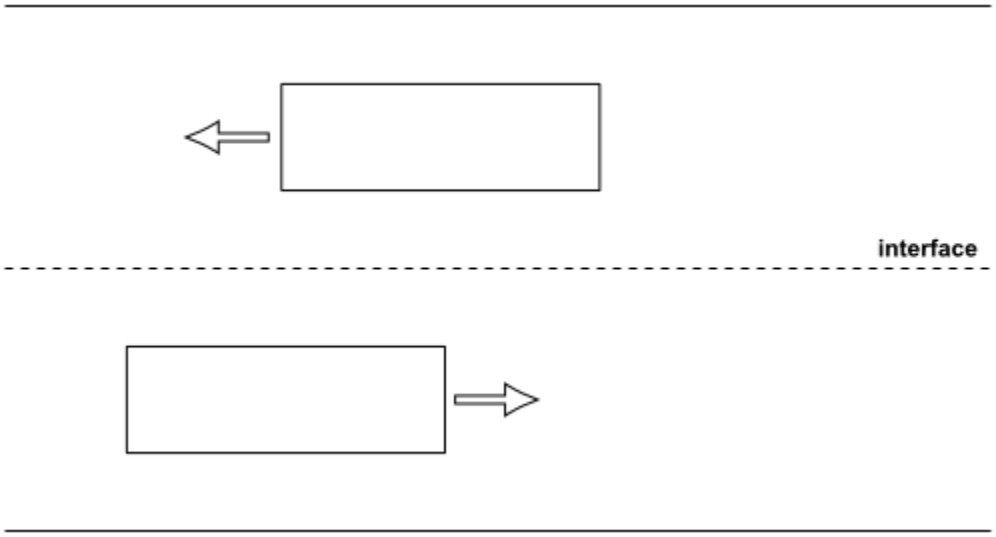
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.2. Sliding Mesh Examples

When a time-accurate solution (rather than a time-averaged solution) for rotor-stator interaction is desired, you must use the sliding mesh model to compute the unsteady flow field. The sliding mesh model is the most accurate method for simulating flows in multiple moving reference frames, but also the most computationally demanding.

Most often, the unsteady solution that is sought in a sliding mesh simulation is time-periodic. That is, the unsteady solution repeats with a period related to the speeds of the moving domains. However, you can model other types of transients, including translating sliding mesh zones (for example, two cars or trains passing in a tunnel, as shown in Figure 10.1: Two Passing Trains in a Tunnel (p. 628)). Note that the sliding mesh can be modeled using periodic boundaries (Figure 10.2: Rotor-Stator Interaction (Stationary Guide Vanes with Rotating Blades) (p. 629)) or a circular/cylindrical interface (Figure 10.3: Blower (p. 629)).

Figure 10.1: Two Passing Trains in a Tunnel



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.2: Rotor-Stator Interaction (Stationary Guide Vanes with Rotating Blades)

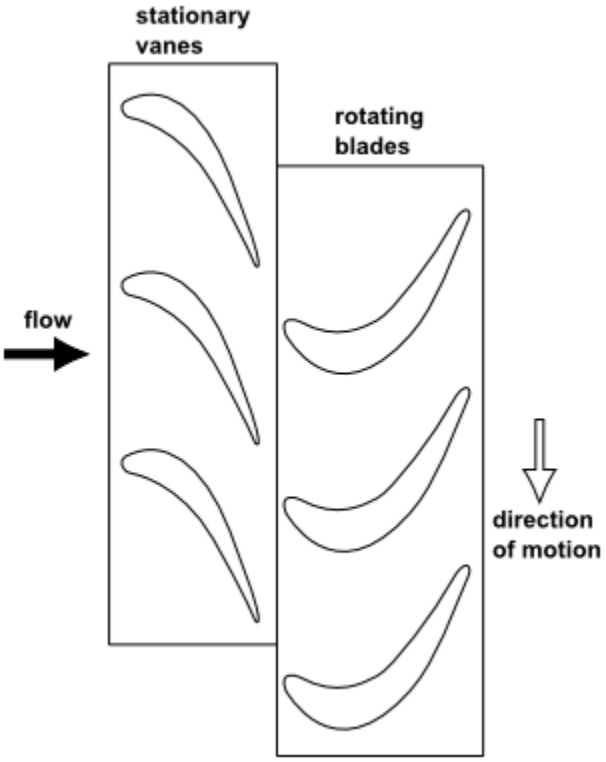
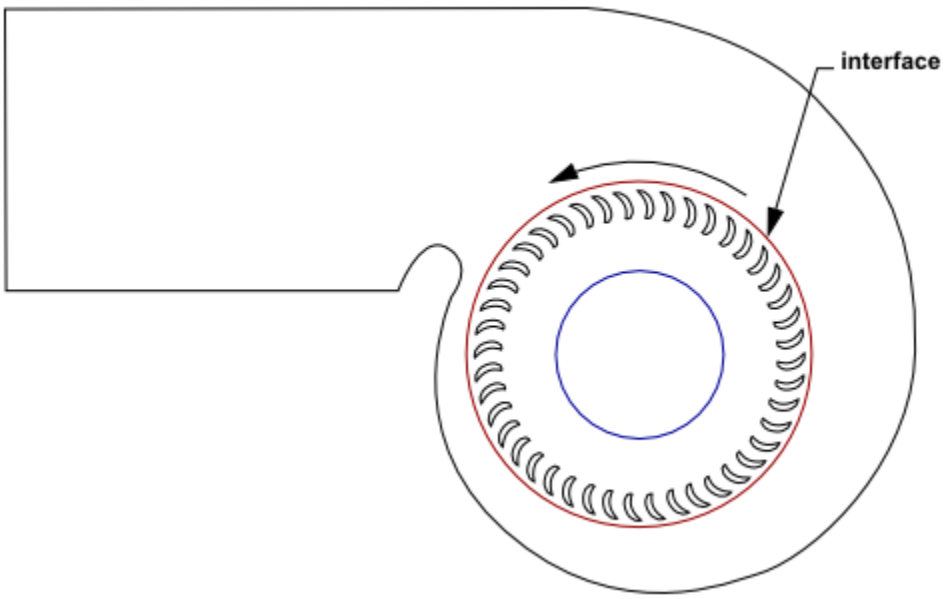


Figure 10.3: Blower



Note that for flow situations where there is no interaction between stationary and moving parts (for example, when there is only a rotor), it is not necessary to use a sliding mesh, and the moving reference frame model is recommended. (See Flow in a Moving Reference Frame for details.) If you are interested in a steady approximation of the interaction, you may use the multiple reference frame model or the mixing plane model, as described in The Multiple Reference Frame Model and The Mixing Plane Model in the Theory Guide.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

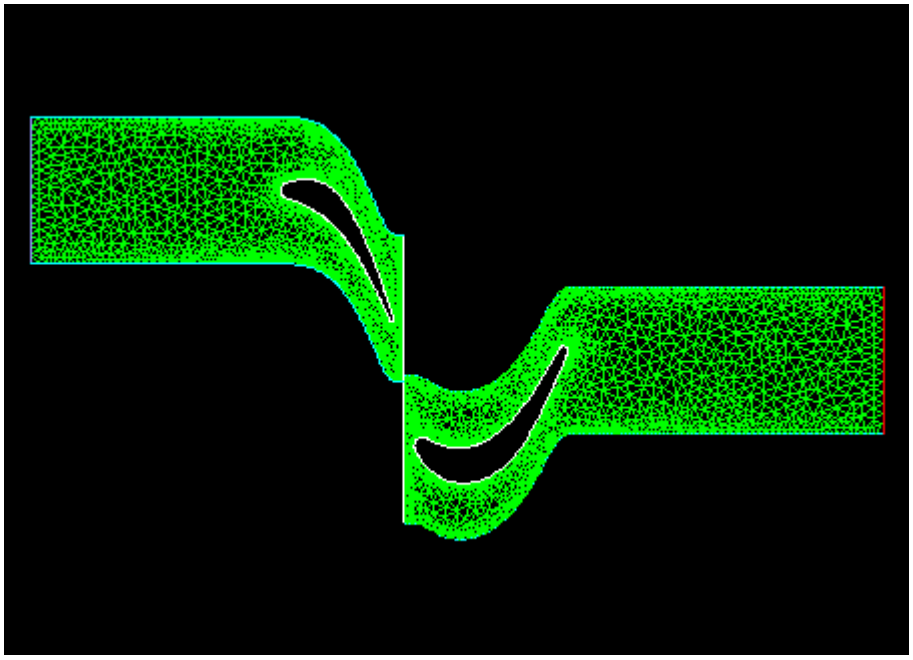
10.3. The Sliding Mesh Technique

In the sliding mesh technique, two or more cell zones are used. (If you generate the mesh in each zone independently, you must merge the mesh files prior to starting the calculation, as described in Reading Multiple Mesh/Case/Data Files in the User's Guide.) Each cell zone is bounded by at least one "interface zone" where it meets the opposing cell zone. The interface zones of adjacent cell zones are associated with one another to form a "mesh interface." The two cell zones will move relative to each other along the mesh interface.

During the calculation, the cell zones slide (that is, rotate or translate) relative to one another along the mesh interface in discrete steps. Figure 10.4: Initial Position of the Meshes (p. 630) and Figure 10.5: Rotor Mesh Slides with Respect to the Stator (p. 631) show the initial position of two meshes and their positions after some translation has occurred. Note that all non-conformal interfaces will be automatically updated (if necessary) by ANSYS Fluent when the mesh is updated.

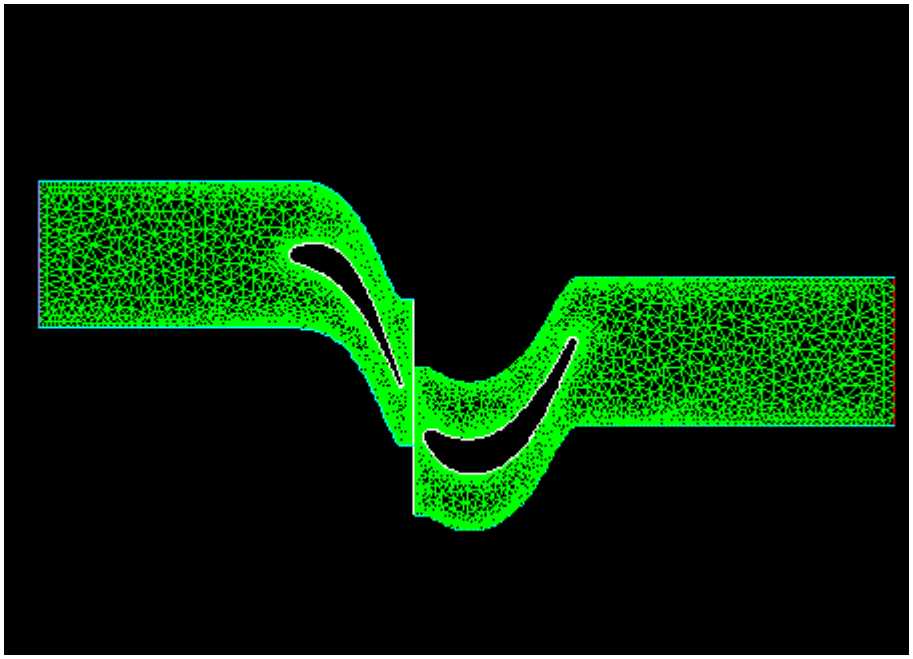
As the rotation or translation takes place, node alignment along the mesh interface is not required. Since the flow is inherently unsteady, a time-dependent solution procedure is required.

Figure 10.4: Initial Position of the Meshes



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

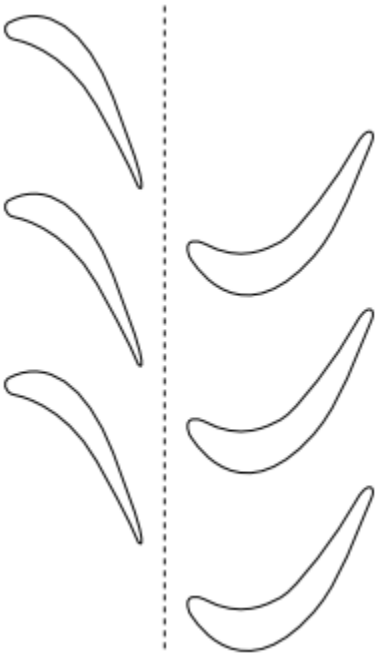
Figure 10.5: Rotor Mesh Slides with Respect to the Stator



10.4. Sliding Mesh Interface Shapes

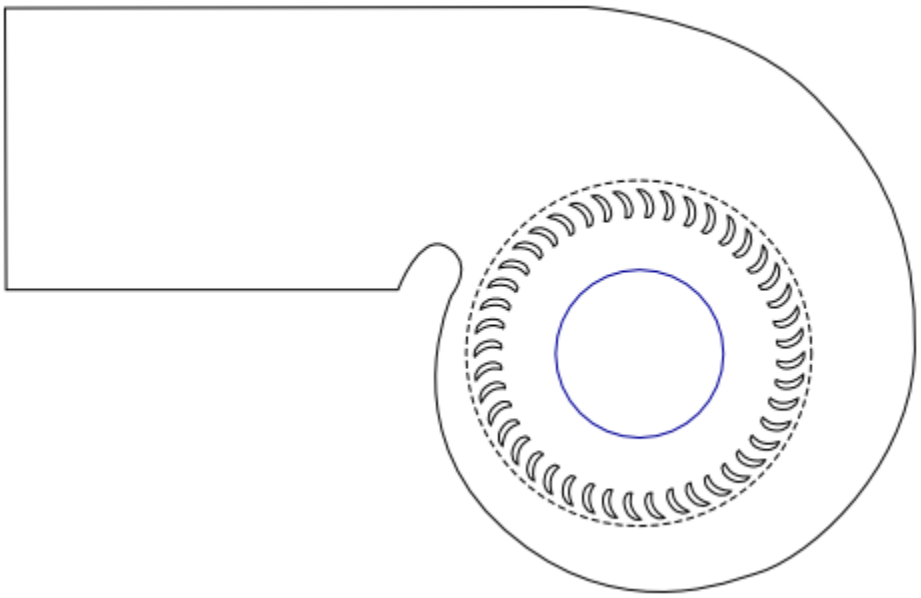
The mesh interface and the associated interface zones can be any shape, provided that the two interface boundaries are based on the same geometry. Figure 10.6: 2D Linear Mesh Interface (p. 631) shows an example with a linear mesh interface and Figure 10.7: 2D Circular-Arc Mesh Interface (p. 632) shows a circular-arc mesh interface. (In both figures, the mesh interface is designated by a dashed line.)

Figure 10.6: 2D Linear Mesh Interface



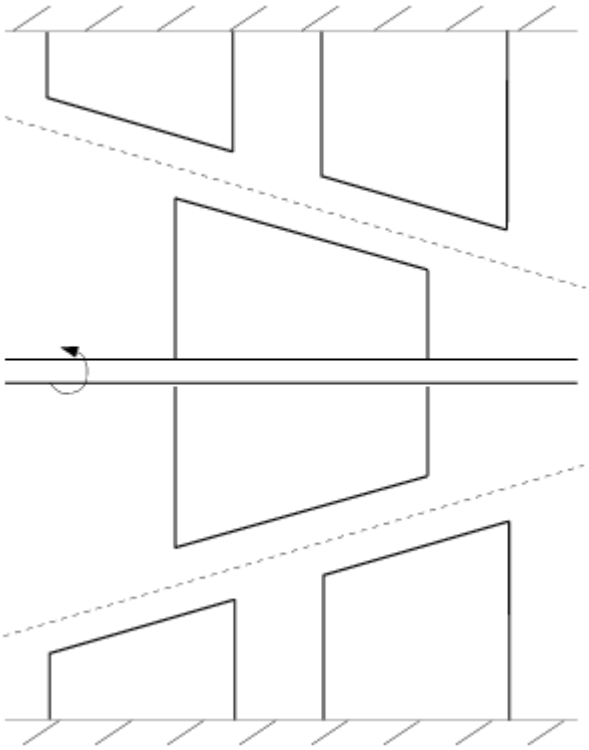
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.7: 2D Circular-Arc Mesh Interface



If Figure 10.6: 2D Linear Mesh Interface (p. 631) was extruded to 3D, the resulting sliding interface would be a planar rectangle; if Figure 10.7: 2D Circular-Arc Mesh Interface (p. 632) was extruded to 3D, the resulting interface would be a cylinder. Figure 10.8: 3D Conical Mesh Interface (p. 632) shows an example that would use a conical mesh interface. (The slanted, dashed lines represent the intersection of the conical interface with a 2D plane.)

Figure 10.8: 3D Conical Mesh Interface

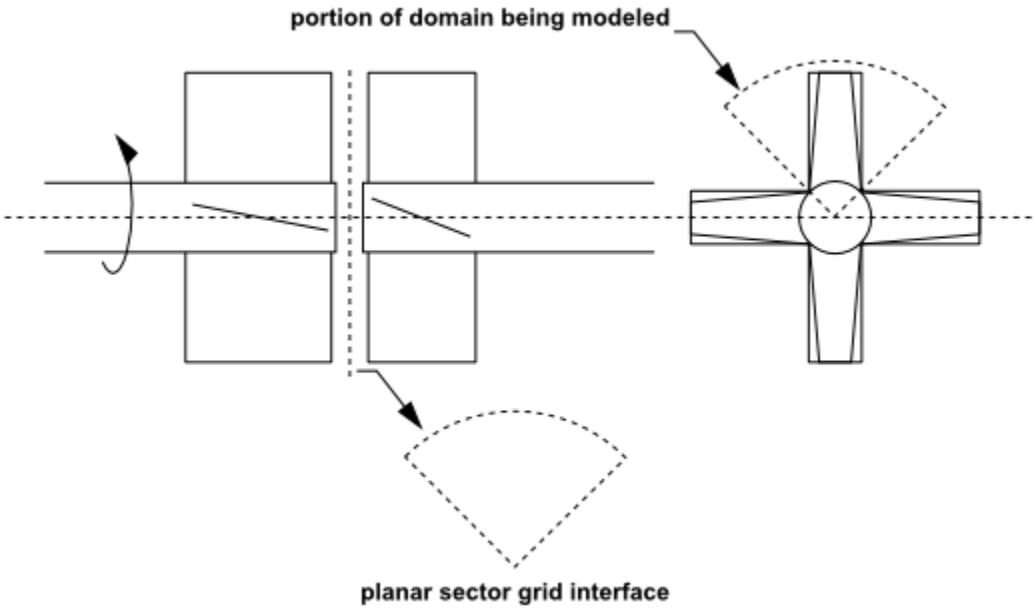


For an axial rotor/stator configuration, in which the rotating and stationary parts are aligned axially instead of being concentric (see Figure 10.9: 3D Planar-Sector Mesh Interface (p. 633)), the interface will be a

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

planar sector. This planar sector is a cross-section of the domain perpendicular to the axis of rotation at a position along the axis between the rotor and the stator.

Figure 10.9: 3D Planar-Sector Mesh Interface



10.5. Using Sliding Meshes

This section describes how to use sliding meshes, including restrictions and constraints, problem setup, solution strategies, and postprocessing.

10.5.1. Requirements, Constraints, and Considerations
10.5.2. Setting Up the Sliding Mesh Problem
10.5.3. Solution Strategies for Sliding Meshes
10.5.4. Postprocessing for Sliding Meshes

10.5.1. Requirements, Constraints, and Considerations

Before beginning the problem setup in ANSYS Fluent, be sure that the mesh you have created meets the following requirements:

- The mesh must have a different cell zone for each portion of the domain that is sliding at a different speed.
- The mesh interface must be situated such that there is no motion normal to it.
- The mesh interface can be any shape (including a non-planar surface, in 3D), provided that the two interface boundaries are based on the same geometry. If there are sharp features in the mesh (for example, 90-degree angles), it is especially important that both sides of the interface closely follow that feature.
- If you create a single mesh with multiple cell zones, you must be sure that each cell zone has a distinct face zone on the sliding boundary. The face zones for two adjacent cell zones will have the same position and shape, but one will correspond to one cell zone and one to the other. (Note that it is also possible to create a separate mesh file for each of the cell zones, and then merge them as described in Reading Multiple Mesh/Case/Data Files (p. 145).)
- If you are modeling a rotor/stator geometry using periodicity, the periodic angle of the mesh around the rotor blade(s) must be the same as that of the mesh around the stationary vane(s).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- All periodic zones must be correctly oriented (either rotational or translational) before you create the mesh interface.
- Note the following limitations if you want to use the periodic repeats option as part of the mesh interface:
  - The edges of the second interface zone must be offset from the corresponding edges of the first interface zone by a uniform amount (either a uniform translational displacement or a uniform rotation angle).
  - Some portion of the two interface zones must overlap (that is, be spatially coincident).
  - The non-overlapping portions of the interface zones must have identical shape and dimensions at all times during the mesh motion.
  - One pair of conformal periodic zones must be adjacent to each of the interface zones. For example, when you calculate just one channel and blade of a fan, turbine, and so on, you must have conformal periodics on either side of the interface threads. This will not work with non-conformal periodics.

    Note that for 3D cases, you cannot have more than one pair of conformal periodic zones adjacent to each of the interface zones.
- You must not have a single sliding mesh interface where part of the interface is made up of a coupled two-sided wall, while another part is not coupled (that is, the normal interface treatment). In such cases, you must break the interface up into two interfaces: one that is a coupled interface, and the other that is a standard fluid-fluid interface. See Using a Non-Conformal Mesh in ANSYS Fluent (p. 164) for information about creating coupled interfaces.
- When using the sliding mesh technique, mass fluxes are interpolated from the sliding interface faces to the sliding boundary faces, which can affect the local mass balance at the sliding boundary. This can result in pressure field oscillations at the sliding boundary. These oscillations may be reduced by ensuring the surface mesh node distributions are similar on either side of the sliding boundaries. You can use the matching option (Matching Option (p. 157)), where applicable, to reduce solution oscillations at the sliding boundaries.

For details about these restrictions and general information about how the sliding mesh model works in ANSYS Fluent, see The Sliding Mesh Technique (p. 630).

10.5.2. Setting Up the Sliding Mesh Problem

The steps for setting up a sliding mesh problem are listed below. (Note that this procedure includes only those steps necessary for the sliding mesh model itself; you must set up other models, boundary conditions, and so on, as usual.)

1. Enable the appropriate option for modeling transient flow. (See Performing Time-Dependent Calculations (p. 1608) for details about the transient modeling capabilities in ANSYS Fluent.)

   Setup → **General** **Analysis Type** → **Transient**

2. Set the cell zone conditions for the sliding motion:

   Setup → **Cell Zone Conditions**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the Solid Dialog Box (p. 2379) or Fluid Dialog Box (p. 2369) of each moving fluid or solid zone, enable the **Mesh Motion** option and set the translational and/or rotational velocity under the **Mesh Motion** tab. (Note that a solid zone cannot move at a different speed than an adjacent fluid zone.)

Important

Note that simultaneous translation and rotation can be modeled only if the rotation axis and the translation direction are the same (that is, the origin is fixed).

Note that the speed can be specified as a constant value or a transient profile. The transient profile may be in a file format, as described in Defining Transient Cell Zone and Boundary Conditions (p. 448), or a UDF macro, described in `DEFINE_TRANSIENT_PROFILE` in the Fluent Customization Manual. Specifying the individual velocities as either a profile or a UDF allows you to specify a single component of the frame motion individually. However, you can also specify the frame motion using a user-defined function. This may prove to be quite convenient if you are modeling a more complicated motion of the moving reference frame, where the hooking of many different user-defined functions or profiles can be cumbersome.

Note

If you decide to hook a UDF, then you will no longer have access to the rotation axis origin and direction, or the velocities.

3. Set the boundary conditions for the sliding motion:

   Setup → Boundary Conditions

   Change the zone **Type** of the interface zones of adjacent cell zones to **interface** in the Boundary Conditions Task Page (p. 2389).

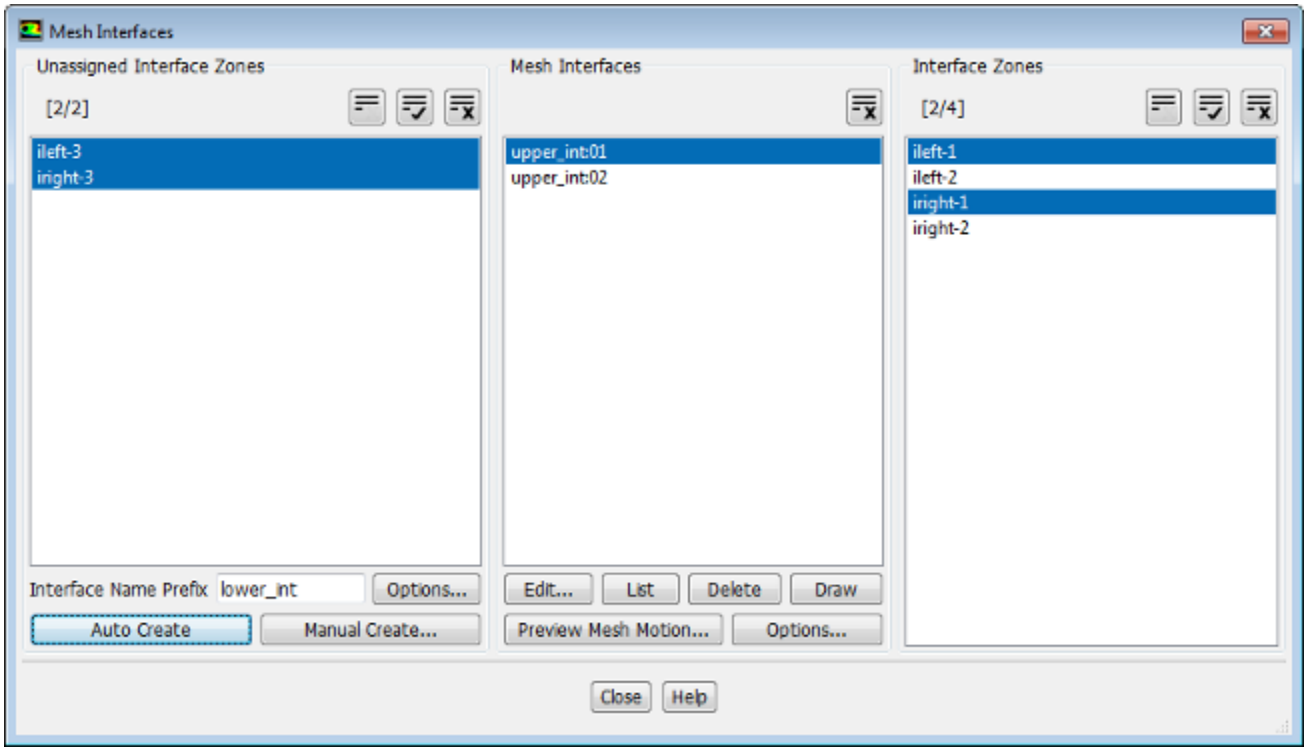
   By default, the velocity of a wall is set to zero relative to the motion of the adjacent mesh. For walls bounding a moving mesh this results in a "no-slip" condition in the reference frame of the mesh. Therefore, you need not modify the wall velocity boundary conditions unless the wall is stationary in the absolute frame, and therefore moving in the relative frame. See Velocity Conditions for Moving Walls (p. 359) for details about wall motion.

   See Cell Zone and Boundary Conditions (p. 237) for details about inputting cell zone and boundary conditions.

4. Define the mesh interfaces using the Mesh Interfaces Dialog Box (p. 2732) (Figure 10.10: The Mesh Interfaces Dialog Box (p. 636)).

   Setup → Mesh Interfaces → New...

Figure 10.10: The Mesh Interfaces Dialog Box



To learn how to use the **Mesh Interfaces** dialog box, see Using a Non-Conformal Mesh in ANSYS Fluent (p. 164). Note that the **Interface Options** that are relevant for sliding meshes are **Periodic Repeats**, **Coupled Wall**, and **Matching** (for more information on these options, see Non-Conformal Mesh Calculations (p. 150)).

5. Preview the mesh motion using the **Zone Motion** dialog box, which can be opened from the **Run Calculation** task page.

   Solution → **Run Calculation** → **Preview Mesh Motion...**

   For details about using the **Zone Motion** dialog box, see Previewing the Dynamic Mesh (p. 742).

Important

When you have completed the problem setup, you should save an initial case file so that you can easily return to the original mesh position (that is, the positions before any sliding occurs). The mesh position is stored in the case file, so case files that you save at different times during the transient calculation will contain meshes at different positions.

Important

If you want to go from an MRF model setup to a sliding mesh setup, use the following text command:

`mesh` → `modify-zones` → `mrf-to-sliding-mesh`

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To successfully switch from an MRF to a sliding mesh, you must provide the ID of the fluid zone. ANSYS Fluent identifies all the zones belonging to this fluid zone, as well as fluid zones shared in the domain. ANSYS Fluent then splits these zones into walls, after which the walls are slit converted to interfaces. ANSYS Fluent then changes the cell zone condition of the fluid zone to **Moving Mesh** in the Fluid Dialog Box (p. 2369). Note that the interface between the cells zones should be an interior boundary. You do not need to do this if you have already created a mesh interface. The sliding mesh solution tends to be more robust than the MRF solution.

10.5.3. Solution Strategies for Sliding Meshes

You will begin the sliding mesh calculation by initializing the solution (as described in Initializing the Entire Flow Field Using Standard Initialization (p. 1591)) and then specifying the time step size and number of time steps in the Run Calculation Task Page (p. 2537), as for any other transient calculation. (See Performing Time-Dependent Calculations (p. 1608) for details about time-dependent solutions.) Note that the time step size in the initial case file is saved without clicking **Calculate**. ANSYS Fluent will iterate on the current time step solution until satisfactory residual reduction is achieved, or the maximum number of iterations per time step is reached. When it advances to the next time step, the cell and wall zones will automatically be moved according to the specified translational or rotational velocities (as discussed in the previous section). The new interface-zone intersections will be computed automatically, and resultant interior/periodic/external boundary zones will be updated.

Note that you can run the MRF case using mesh interfaces with an appreciable loss of accuracy, doing so makes it easier to later on convert to a sliding mesh.

Note

The sliding mesh can be initialized with an MRF solution (rather using Standard or Hybrid initializations).

It is recommended that you preview the sliding mesh motion (as described in Previewing the Dynamic Mesh (p. 742)) before beginning your calculation. This can catch problems with the motion specification(s) before you begin the CFD calculation. Remember to save the case and initial data files before doing a mesh preview since the mesh position is altered once you do the preview. You can reread the initial condition case/data files to get back to the original mesh position.

10.5.3.1. Saving Case and Data Files

ANSYS Fluent's automatic saving of case and data files (see Automatic Saving of Case and Data Files (p. 48)) can be used with the sliding mesh model. This provides a convenient way for you to save results at successive time steps for later postprocessing.

Important

You must save a case file each time you save a data file because the mesh position is stored in the case file. Since the mesh position changes with each time step, reading data for a given time step will require the case file at that time step so that the mesh will be in the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

proper position. You should also save your initial case file so that you can easily return to the mesh's original position to restart the solution if desired.

Important

If you are planning to solve your sliding mesh model in several stages, whereby you run the calculation for some period of time, save case and data files, exit ANSYS Fluent, start a new ANSYS Fluent session, read the case and data files, continue the calculation for some time, save case and data files, exit ANSYS Fluent, and so on, there may be some distortion in the mesh with each subsequent continuation of the calculation. To avoid this problem, you can delete the mesh interface before saving the case file, and then create it again after you read the case file into a new ANSYS Fluent session.

10.5.3.2. Time-Periodic Solutions

For some problems (for example, rotor-stator interactions), you may be interested in a time-periodic solution. That is, the startup transient behavior may not be of interest to you. Once this startup phase has passed, the flow will start to exhibit time-periodic behavior. If T is the period of unsteadiness, then for some flow property ϕ at a given point in the flow field:

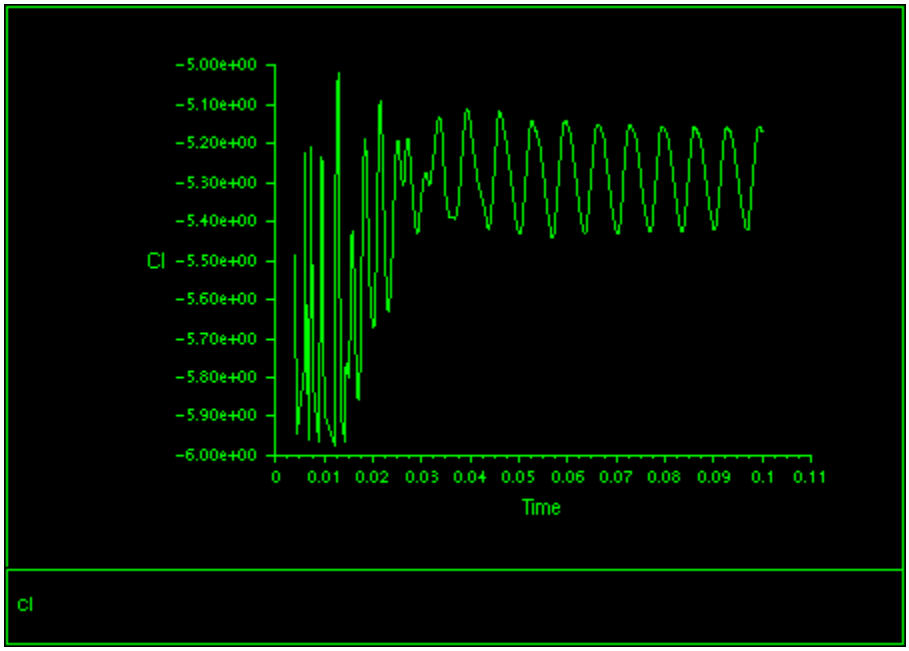
$$\phi(t)=\phi(t+NT)(N=1,2,3,\ldots) \tag{10.1}$$

For rotating problems, the period (in seconds) can be calculated by dividing the sector angle of the domain (in radians) by the rotor speed (in radians/sec): $T=\theta/\Omega$. For 2D rotor-stator problems, $T=P/v_b$, where P is the pitch and v_b is the blade speed. The number of time steps in a period can be determined by dividing the time period by the time step size. When the solution field does not change from one period to the next (for example, if the change is less than 5%), a time-periodic solution has been reached.

To determine how the solution changes from one period to the next, you must compare the solution at some point in the flow field over two periods. For example, if the time period is 10 seconds, you can compare the solution at a given point after 22 seconds with the solution after 32 seconds to see if a time-periodic solution has been reached. If not, you can continue the calculation for another period and compare the solutions after 32 and 42 seconds, and so on until you see little or no change from one period to the next. You can also track global quantities, such as lift and drag coefficients and mass flow, in the same manner. Figure 10.11: Lift Coefficient Plot for a Time-Periodic Solution (p. 639) shows a lift coefficient plot for a time-periodic solution.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.11: Lift Coefficient Plot for a Time-Periodic Solution



The final time-periodic solution is independent of the time steps taken during the initial stages of the solution procedure. You can therefore define "large" time steps in the initial stages of the calculation, since you are not interested in a time-accurate solution for the startup phase of the flow. Starting out with large time steps will allow the solution to become time-periodic more quickly. As the solution becomes time-periodic, however, you should reduce the time step in order to achieve a time-accurate result.

Important

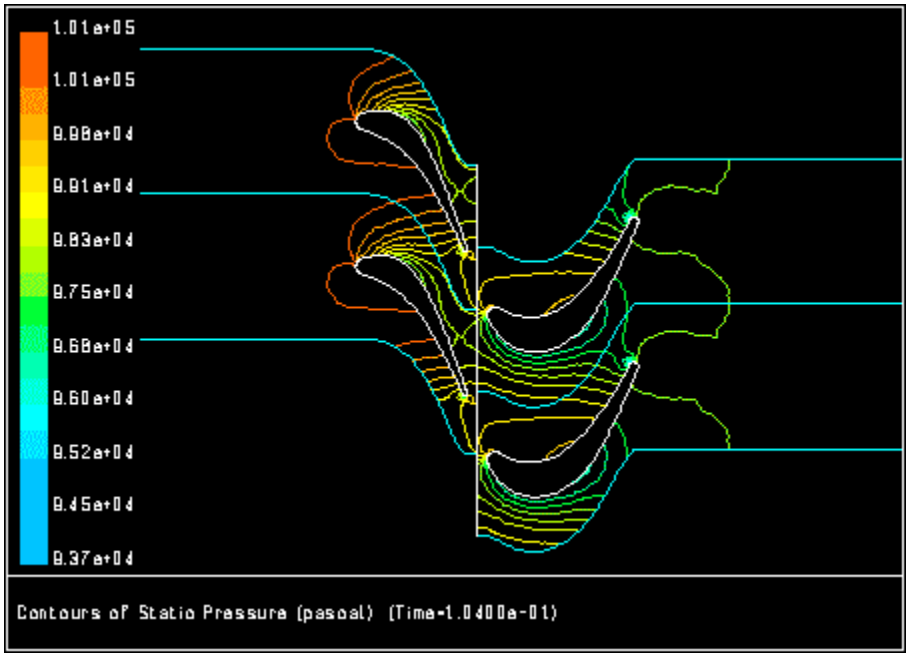
If you are solving with second-order time accuracy, the temporal accuracy of the solution will be affected if you change the time step during the calculation. You may start out with larger time steps, but you should not change the time step by more than 20% during the solution process. You should not change the time step at all during the last several periods to ensure that the solution has approached a time-periodic state.

10.5.4. Postprocessing for Sliding Meshes

Postprocessing for sliding mesh problems is the same as for other transient problems. You will read in the case and data file for the time of interest and display and report results as usual. For spatially-periodic problems, you may want to use periodic repeats (set in the Views Dialog Box (p. 2583), as described in Modifying the View (p. 1803)) to display the geometry. Figure 10.12: Contours of Static Pressure for the Rotor-Stator Example (p. 640) shows the flow field for the rotor-stator example in Figure 10.4: Initial Position of the Meshes (p. 630)) at one instant in time, using 1 periodic repeat.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.12: Contours of Static Pressure for the Rotor-Stator Example



When displaying velocity vectors, note that absolute velocities (that is, velocities in the inertial, or laboratory, reference frame) are displayed by default. You may also choose to display relative velocities by selecting **Relative Velocity** in the **Vectors of** drop-down list in the Vectors Dialog Box (p. 2831). In this case, velocities relative to the translational/rotational velocity of the "reference zone" (specified in the Reference Values Task Page (p. 2502)) will be displayed. (The velocity of the reference zone is the velocity defined in the Fluid Dialog Box (p. 2369) for that zone.)

Note that you cannot create zone surfaces for the intersection boundaries (that is, the interior/periodic/external zones created from the intersection of the interface zones). You may instead create zone surfaces for the interface zones. Data displayed on these surfaces will be "one-sided". That is, nodes on the interface zones will "see" only the cells on one side of the mesh interface, and slight discontinuities may appear when you plot contour lines across the interface. Note also that, for non-planar interface shapes in 3D, you may see small gaps in your plots of filled contours. These discontinuities and gaps are only graphical in nature. The solution does not have these discontinuities or gaps. To eliminate these discontinuities for postprocessing purposes only, you can use the `define/mesh-interfaces/enforce-continuity-after-bc?` text command, which will ensure that continuity will take precedence over the boundary condition.

You can also generate a plot of circumferential averages in ANSYS Fluent. This allows you to find the average value of a quantity at several different radial or axial positions in your model. ANSYS Fluent computes the average of the quantity over a specified circumferential area, and then plots the average against the radial or axial coordinate. For more information on generating XY plots of circumferential averages, see XY Plots of Circumferential Averages (p. 1848).

Sliding mesh results can be analyzed by employing the time averaging (or RMSE averaging) option (**Data Sampling for Time Statistics**) in the **Run Calculation** task page. This will compute time averages for velocity, pressure, temperature, and turbulence. You must plan ahead since you have to engage the time averaging after the solution has become time-periodic and run for at least one period of the oscillating flow field.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.6. Using Dynamic Meshes

The steps for setting up a dynamic mesh problem are listed below. (Note that this procedure includes only those steps necessary for the dynamic mesh model itself; you must set up other models, cell zone conditions, boundary conditions, and so on, as usual.)

1. Enable the appropriate option for modeling transient or steady flow in the General Task Page (p. 2160).

   Setup → General

   If your problem involves a steady flow, see Steady-State Dynamic Mesh Applications (p. 744) for important considerations.

2. Set cell zone conditions and boundary conditions as required in the Cell Zone Conditions Task Page (p. 2367) and Boundary Conditions Task Page (p. 2389).

   Setup → Cell Zone Conditions

   Setup → Boundary Conditions

   See Cell Zone and Boundary Conditions (p. 237) for information about inputting conditions. The correct wall velocity is set up automatically when a dynamic zone is created for a wall zone and the motion attributes are specified, so you will not specify wall motion in the **Wall** dialog box. If you create a moving dynamic cell zone, then all wall boundaries adjacent to that cell zone will, by default, impose the correct (moving) boundary conditions, and it is not necessary to declare these wall zones as dynamic zones. Note that if a wall boundary mesh is moving because it belongs to an adjacent moving dynamic cell zone, but the physical boundary conditions are such that the wall is actually not moving, then you will have to declare this boundary as a dynamic zone and specify that the mesh motion is not included in the boundary conditions (see Specifying the Motion of Dynamic Zones (p. 722)).

3. Enable the dynamic mesh model, and specify related parameters in the Dynamic Mesh Task Page (p. 2471).

   Setup → Dynamic Mesh → Dynamic Mesh

   See Setting Dynamic Mesh Modeling Parameters (p. 643) for details.

4. Create the dynamic zones for your model, using the Dynamic Mesh Zones Dialog Box (p. 2490).

   Setup → Dynamic Mesh → Create/Edit...

   See Specifying the Motion of Dynamic Zones (p. 722) for details.

5. You can display the motion of the moving zones with prescribed motion to verify the simulation setup.

   Setup → Dynamic Mesh → Display Zone Motion...

   See Previewing the Dynamic Mesh (p. 742) for details.

6. If it is a transient simulation, define the events that will occur during the calculation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → Dynamic Mesh → Events...

See Defining Dynamic Mesh Events (p. 713) for details.

7. Save the case and data.

File → Write → Case & Data...

8. Preview your dynamic mesh setup (when the motion is a prescribed motion). See Steady-State Dynamic Mesh Applications (p. 744) for previewing your steady-state dynamic mesh motion and refer to Previewing the Dynamic Mesh (p. 742) for details.

Setup → Dynamic Mesh → Preview Mesh Motion...

9. Specify the pressure-velocity coupling scheme. For transient flow calculations, the PISO algorithm is recommended, as it is the most efficient for such cases (see PISO (p. 1562) for details).

10. Use the automatic saving feature to specify the file name and frequency with which case and data files should be saved during the solution process.

Solution → Calculation Activities → Autosave (Every) → Edit...

See Automatic Saving of Case and Data Files (p. 48) for details about the use of this feature. This provides a convenient way for you to save results at successive time steps for later postprocessing.

Important

You must save a case file each time you save a data file because the mesh position is stored in the case file. Since the mesh position changes with each time step, reading data for a given time step will require the case file at that time step so that the mesh will be in the proper position. You should also save your initial case file so that you can easily return to the mesh's original position to restart the solution if desired.

11. (optional) If you want to create a graphical animation of the mesh over time during the solution procedure, you can use the Calculation Activities Task Page (p. 2525) to set up the graphical displays that you want to use in the animation. See Animating the Solution (p. 1647) for details.

For additional information, see the following sections:

10.6.1. Setting Dynamic Mesh Modeling Parameters
10.6.2. Dynamic Mesh Update Methods
10.6.3. Feature Detection
10.6.4. In-Cylinder Settings
10.6.5. Six DOF Solver Settings
10.6.6. Implicit Update Settings
10.6.7. Contact Detection Settings
10.6.8. Defining Dynamic Mesh Events
10.6.9. Specifying the Motion of Dynamic Zones
10.6.10. Previewing the Dynamic Mesh
10.6.11. Steady-State Dynamic Mesh Applications

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.6.1. Setting Dynamic Mesh Modeling Parameters

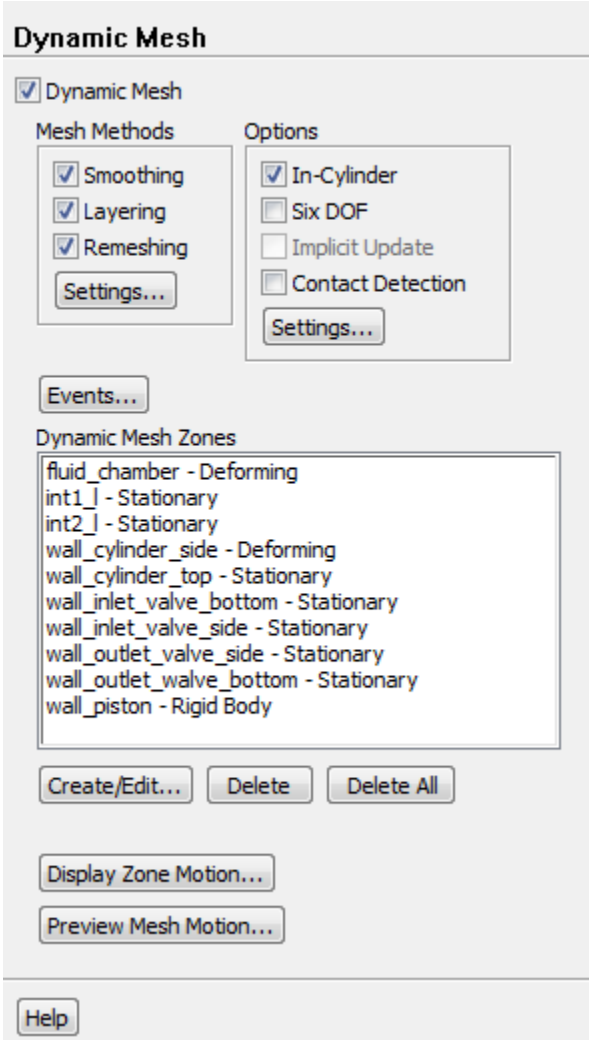
To enable the dynamic mesh model, enable **Dynamic Mesh** in the Dynamic Mesh Task Page (p. 2471) (Figure 10.13: The Dynamic Mesh Task Page (p. 643)).

Setup → **Dynamic Mesh** → **Dynamic Mesh**

Then, enable the appropriate options in the **Options** group box. If you are modeling in-cylinder motion, enable the **In-Cylinder** option. If you are going to use the six degrees of freedom solver, then enable the **Six DOF** option. If you want to have the dynamic mesh updated during a time step (as opposed to just at the beginning of a time step), then enable the **Implicit Update** option. More information about these options and the related settings can be found in In-Cylinder Settings (p. 690), Six DOF Solver Settings (p. 705), and Implicit Update Settings (p. 709), respectively.

Next, you must select the appropriate mesh update methods in the **Mesh Methods** group box, and set the associated parameters, if relevant. See Dynamic Mesh Update Methods (p. 644) for details.

Figure 10.13: The Dynamic Mesh Task Page



10.6.2. Dynamic Mesh Update Methods

Three groups of mesh motion methods are available in ANSYS Fluent to update the volume mesh in the deforming regions subject to the motion defined at the boundaries:

- Smoothing Methods (p. 644)
- Dynamic Layering (p. 661)
- Remeshing Methods (p. 665)

Note that you can use ANSYS Fluent's dynamic mesh models in conjunction with hanging node adaption, with the exception of dynamic layering and face remeshing. For more information on hanging node adaption, see Hanging Node Adaption in the Theory Guide.

Details on how to set up the various dynamic mesh update methods are provided in the sections that follow.

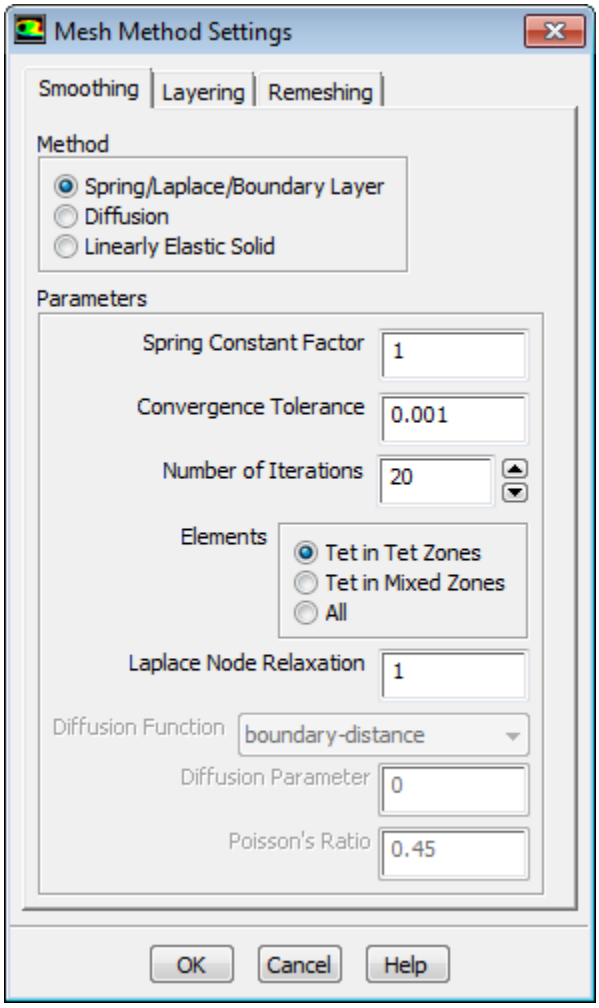
10.6.2.1. Smoothing Methods

When smoothing is used to adjust the mesh of a zone with a moving and/or deforming boundary, the interior nodes of the mesh move, but the number of nodes and their connectivity does not change. In this way, the interior nodes "absorb" the movement of the boundary. To enable smoothing, perform the following steps:

1. Enable the **Smoothing** option in the **Mesh Methods** group box of the Dynamic Mesh Task Page (p. 2471).
2. Click the **Settings...** button to open the **Mesh Method Settings** dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.14: The Smoothing Tab of the Mesh Method Settings Dialog Box (3D)



3. If you want spring-based smoothing, select **Spring/Laplace/Boundary Layer** from the **Method** list and define the settings in the **Parameters** group box, which are described in Spring-Based Smoothing (p. 646).

4. If you want diffusion-based smoothing (described in Diffusion-Based Smoothing (p. 650)):

   a. Select **Diffusion** from the **Method** list.

   b. Make a selection from the **Diffusion Function** drop-down list, to indicate whether you want the diffusion coefficient to be a function of the **boundary-distance** or the **cell-volume**. These functions and suggested values for the **Diffusion Parameter** are described in Diffusivity Based on Boundary Distance (p. 653) and Diffusivity Based on Cell Volume (p. 655).

5. If you want smoothing using the linearly elastic solid method (described in Linearly Elastic Solid Based Smoothing Method (p. 656)):

   a. Select **Linearly Elastic Solid** from the **Method** list.

   b. Enter the **Poisson's Ratio** in the **Parameters** group box.

6. If you selected **Diffusion** or **Linearly Elastic Solid** smoothing and the simulation involves periodic or quasi-periodic motion, you can specify that the smoothing uses a reference position by using the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

following text command. This option may improve the mesh quality consistency from cycle to cycle—see Smoothing from a Reference Position (p. 657) for details.

`define` → `dynamic-mesh` → `controls` → `smoothing-parameters` → `smooth-from-reference-position?`

7. If you plan to apply the 2.5D remeshing method (as described in 2.5D Surface Remeshing Method (p. 685)), perform the following steps to set up Laplacian smoothing (as described in Laplacian Smoothing Method (p. 657)).

   a. Select **Spring/Laplace/Boundary Layer** from the **Method** list

   b. Define only the **Laplace Node Relaxation** and the **Number of Iterations** in the **Parameters** group box (the other settings are not relevant).

8. If you plan to apply the boundary layer smoothing method (as described in Boundary Layer Smoothing Method (p. 658)), select **Spring/Laplace/Boundary Layer** from the **Method** list.

10.6.2.1.1. Spring-Based Smoothing

For spring-based smoothing, the edges between any two mesh nodes are idealized as a network of interconnected springs. The initial spacings of the edges before any boundary motion constitute the equilibrium state of the mesh. A displacement at a given boundary node will generate a force proportional to the displacement along all the springs connected to the node. Using Hook's Law, the force on a mesh node can be written as

$$\vec{F}_i = \sum_j^{n_i} k_{ij}\left(\Delta\vec{x}_j - \Delta\vec{x}_i\right) \tag{10.2}$$

where $\Delta\vec{x}_i$ and $\Delta\vec{x}_j$ are the displacements of node i and its neighbor j, n_i is the number of neighboring nodes connected to node i, and k_{ij} is the spring constant (or stiffness) between node i and its neighbor j. The spring constant for the edge connecting nodes i and j is defined as

$$k_{ij} = \frac{k_{\text{fac}}}{\sqrt{\left|\vec{x}_i - \vec{x}_j\right|}} \tag{10.3}$$

where k_{fac} is the value you enter for **Spring Constant Factor**.

At equilibrium, the net force on a node due to all the springs connected to the node must be zero. This condition results in an iterative equation such that

$$\Delta\vec{x}_i^{m+1} = \frac{\sum_j^{n_i} k_{ij}\Delta\vec{x}_j^m}{\sum_j^{n_i} k_{ij}} \tag{10.4}$$

where m is the iteration number.

Since displacements are known at the boundaries (after boundary node positions have been updated), Equation 10.4 (p. 646) is solved using a Jacobi sweep on all interior nodes. At convergence, the positions are updated such that

$$\vec{x}_i^{n+1} = \vec{x}_i^n + \Delta\vec{x}_i^{\text{converged}} \tag{10.5}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where $n+1$ and n are used to denote the positions at the next time step and the current time step, respectively. The spring-based smoothing is shown in Figure 10.15: Spring-Based Smoothing on Interior Nodes: Start (p. 647) and Figure 10.16: Spring-Based Smoothing on Interior Nodes: End (p. 647) for a cylindrical cell zone where one end of the cylinder is moving.

Figure 10.15: Spring-Based Smoothing on Interior Nodes: Start

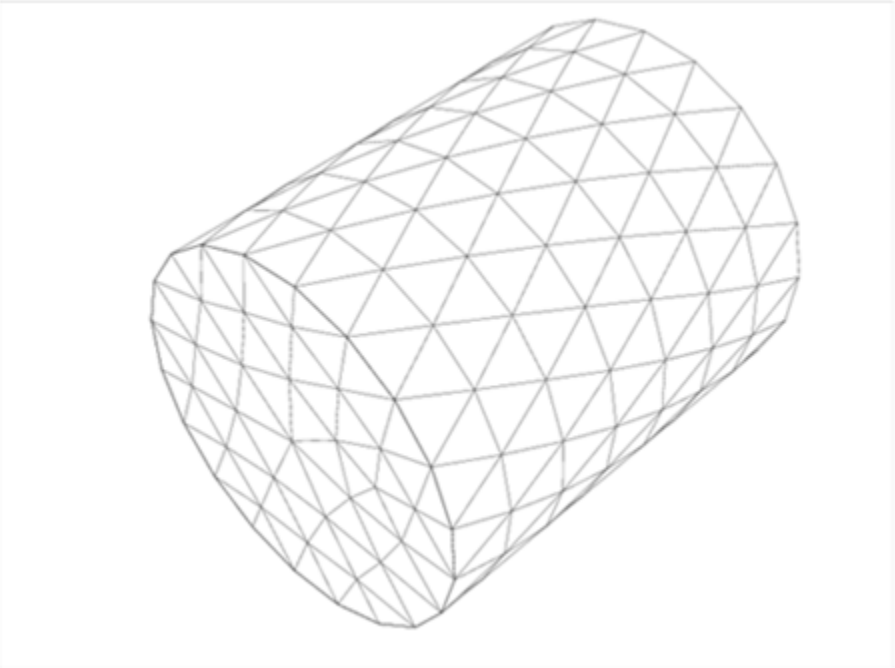
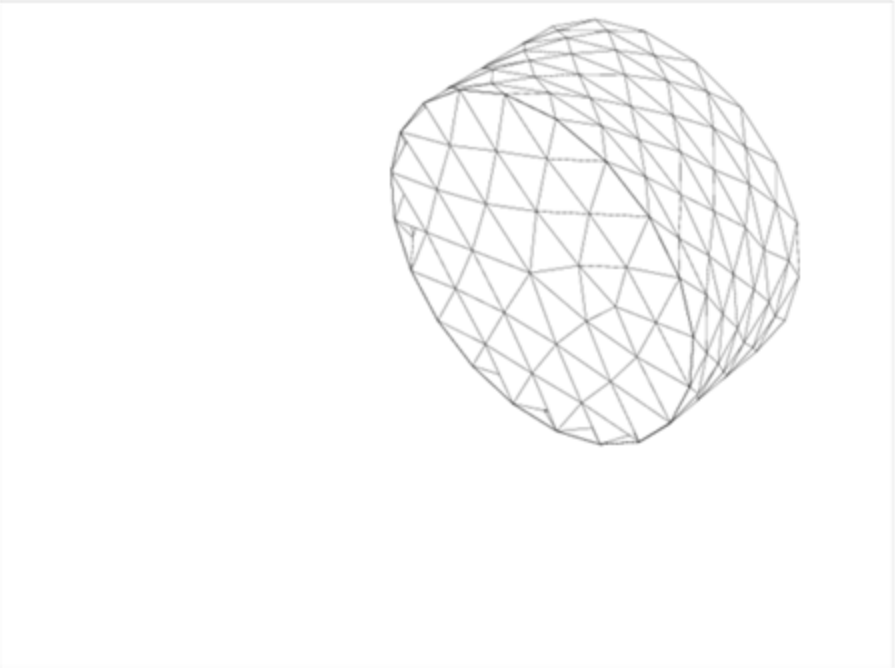


Figure 10.16: Spring-Based Smoothing on Interior Nodes: End

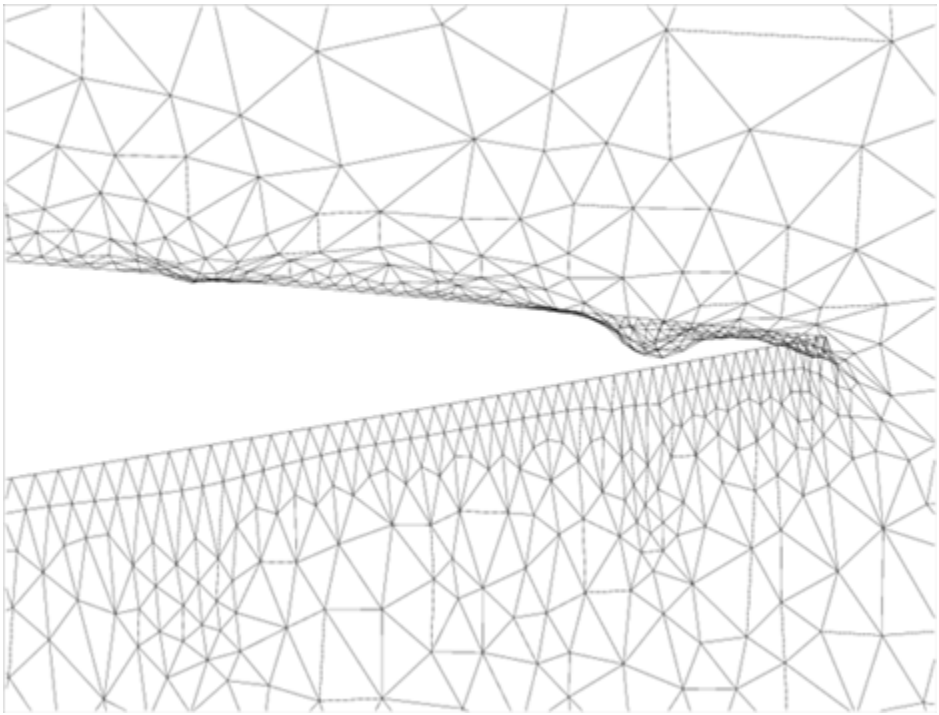


You can control the spring stiffness by adjusting the value of the **Spring Constant Factor** between 0 and 1. A value of 0 indicates that there is no damping on the springs, and boundary node displacements have more influence on the motion of the interior nodes. A value of 1 imposes the default level of damping on the interior node displacements as determined by solving Equation 10.4 (p. 646).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

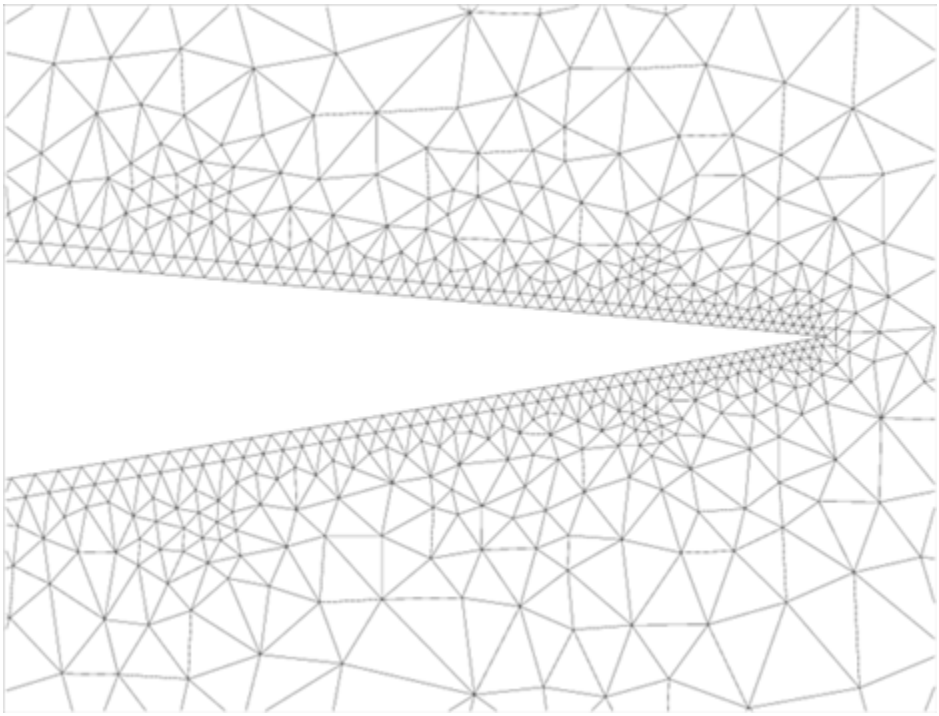
The effect of the **Spring Constant Factor** is illustrated in Figure 10.17: Interior Nodes Extend Beyond Boundary (Spring Constant Factor = 1) (p. 648) and Figure 10.18: Interior Nodes Remain Within Boundary (Spring Constant Factor = 0) (p. 649), which show the trailing edge of a NACA-0012 airfoil after a counter-clockwise rotation of 2.3° and the mesh is smoothed using the spring-based smoother but limited to 20 iterations. Degenerate cells (Figure 10.17: Interior Nodes Extend Beyond Boundary (Spring Constant Factor = 1) (p. 648)) are created with the default value of 1 for the **Spring Constant Factor**, as the interior nodes extend beyond the moving boundary. However, the original mesh distribution (Figure 10.18: Interior Nodes Remain Within Boundary (Spring Constant Factor = 0) (p. 649)) is recovered if the **Spring Constant Factor** is set to 0 (that is, no damping on the displacement of interior nodes near the airfoil surface).

Figure 10.17: Interior Nodes Extend Beyond Boundary (Spring Constant Factor = 1)



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.18: Interior Nodes Remain Within Boundary (Spring Constant Factor = 0)



You can control the solution of Equation 10.4 (p. 646) using the values of **Convergence Tolerance** and **Number of Iterations**. ANSYS Fluent solves Equation 10.4 (p. 646) iteratively during each time step until one of the following criteria is met:

- The specified number of iterations has been performed.
- The solution is converged for that time step:

$$\left(\frac{\Delta\vec{x}^{m}_{rms}}{\Delta\vec{x}^{1}_{rms}}\right) < \text{convergence tolerance} \tag{10.6}$$

  where $\Delta\vec{x}^{1}_{rms}$ is the interior and deforming nodes RMS displacement at the first iteration.

10.6.2.1.1.1. Applicability of the Spring-Based Smoothing Method

You can use the spring-based smoothing method to update any cell or face zone whose boundary is moving or deforming.

For non-tetrahedral cell zones (non-triangular in 2D), the spring-based method can be used when the following conditions are met:

- The boundary of the cell zone moves predominantly in one direction (that is, no excessive anisotropic stretching or compression of the cell zone).
- The motion is predominantly normal to the boundary zone.

If these conditions are not met, the resulting cells may have high skewness values, since not all possible combinations of node pairs in non-tetrahedral cells (or non-triangular in 2D) are idealized as springs. Polyhedral cells are particularly likely to become highly skewed with spring-based smoothing (regardless of whether the previous conditions are met), and so the diffusion-based smoothing method is generally recommended for polyhedra (see Diffusion-Based Smoothing (p. 650)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

By default, spring-based smoothing is disabled for cell zones that are not entirely composed of either tetrahedral or triangular cells. This is reflected in the **Elements** options under **Parameters** in the **Mesh Method Settings** dialog box (Figure 10.14: The Smoothing Tab of the Mesh Method Settings Dialog Box (3D) (p. 645)). By default, the **Elements** are set to **Tet in Tet Zones** in 3D (or **Tri in Tri Zones** in 2D). If you want to use spring-based smoothing on all element types, you can do that by selecting the **All** option.

If you have mixed element zones and you do not want spring-based smoothing on all element types, you can enable spring-based smoothing on only the tetrahedral or triangular cells by selecting **Tet in Mixed Zones** in 3D (or **Tri in Mixed Zones** in 2D). Selection of smoothing elements in the **Mesh Method Settings** dialog box applies by default to all cell zones that undergo spring-based smoothing. In order to have more precise control, it is possible to overwrite this global selection on individual dynamic cell zones (see Deforming Motion (p. 731)). This gives, for example, the flexibility to suppress smoothing in zones where dynamic layering (see Dynamic Layering (p. 661)) is used and allows at the same time smoothing of non-simplex (that is not tetrahedral or triangular) elements in other zones.

10.6.2.1.2. Diffusion-Based Smoothing

For diffusion-based smoothing, the mesh motion is governed by the diffusion equation

$$\nabla \cdot \left(\gamma \nabla \vec{u} \right) = 0 \tag{10.7}$$

where $\vec{u}$ is the mesh displacement velocity. The boundary conditions for Equation 10.7 (p. 650) are obtained from the user-prescribed or computed (six DOF) boundary motion. On deforming boundaries, the boundary conditions are such that the mesh motion is tangent to the boundary (that is, the normal velocity component vanishes). The Laplace equation Equation 10.7 (p. 650) then describes how the prescribed boundary motion diffuses into the interior of the deforming mesh.

The diffusion coefficient γ in Equation 10.7 (p. 650) can be used to control how the boundary motion affects the interior mesh motion. A constant coefficient means that the boundary motion diffuses uniformly throughout the mesh. With a nonuniform diffusion coefficient, mesh nodes in regions with high diffusivity tend to move together (that is, with less relative motion).

In Fluent, two different formulations are available for the diffusion coefficient γ. The first formulation allows you to have the diffusion coefficient be a function of the boundary distance, and is of the form

$$\gamma = \frac{1}{d^{\alpha}} \tag{10.8}$$

where d is a normalized boundary distance. The second formulation allows you to have the diffusion coefficient be a function of the cell volume, and is of the form

$$\gamma = \frac{1}{V^{\alpha}} \tag{10.9}$$

where V is the normalized cell volume. In both Equation 10.8 (p. 650) and Equation 10.9 (p. 650), $\alpha \geq 0$ is a user input parameter. See Diffusivity Based on Boundary Distance (p. 653) and Diffusivity Based on Cell Volume (p. 655) for information about defining the diffusion coefficient.

ANSYS Fluent uses different numerical methods to solve the vector Equation 10.7 (p. 650), depending on the element types present in the mesh. In the absence of polyhedral elements or elements with hanging nodes (that is, adapted and some hexcore or CutCell meshes) the equation is solved using a finite element discretization and the displacement velocity, $\vec{u}$, is obtained directly at each mesh node. If the mesh contains polyhedral elements or hanging nodes, the equation is discretized using ANSYS Fluent's standard finite volume method and the cell-centered solution for the displacement velocity,

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$\vec{u}$, is interpolated onto the nodes using inverse-distance-weighted averaging. The node positions are then updated according to:

$$\vec{x}_{new}=\vec{x}_{old}+\vec{u}\Delta t \tag{10.10}$$

The finite element discretization is generally superior, as the solution is obtained directly at the nodes and no interpolation step is necessary. The finite volume method can be enforced for meshes of all element types by executing the TUI command:

```
/define/dynamic-mesh/controls/smoothing-parameters/diffusion-fvm? yes
```

Computationally, solving a PDE for mesh smoothing is generally more costly than spring-based smoothing. But it tends to produce better quality meshes than spring-based smoothing and often allows larger boundary deformations before breaking down. Figure 10.19: The Initial Mesh (p. 651) and Figure 10.20: Valid Mesh After 45 Degree Rotation Using Diffusion-Based Smoothing (p. 652) show a mesh before and after rotating the boundary by 45 degrees, using diffusion-based smoothing. With spring-based smoothing, the same mesh shows degenerated cells after a rotation of 40 degrees (Figure 10.21: Degenerated Mesh After 40 Degree Rotation Using Spring-Based Smoothing (p. 652)).

Figure 10.19: The Initial Mesh

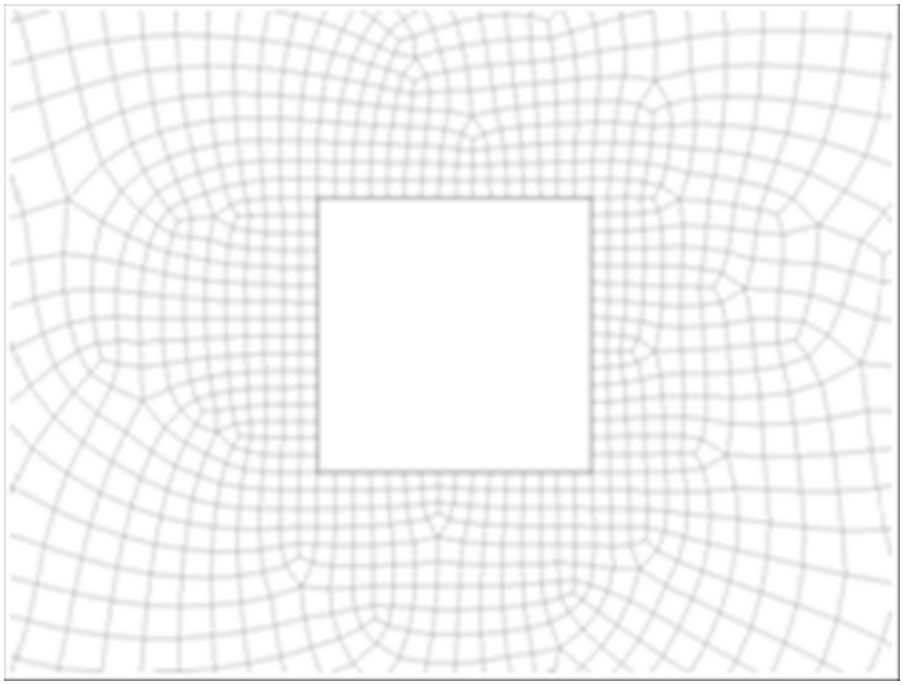


Figure 10.20: Valid Mesh After 45 Degree Rotation Using Diffusion-Based Smoothing

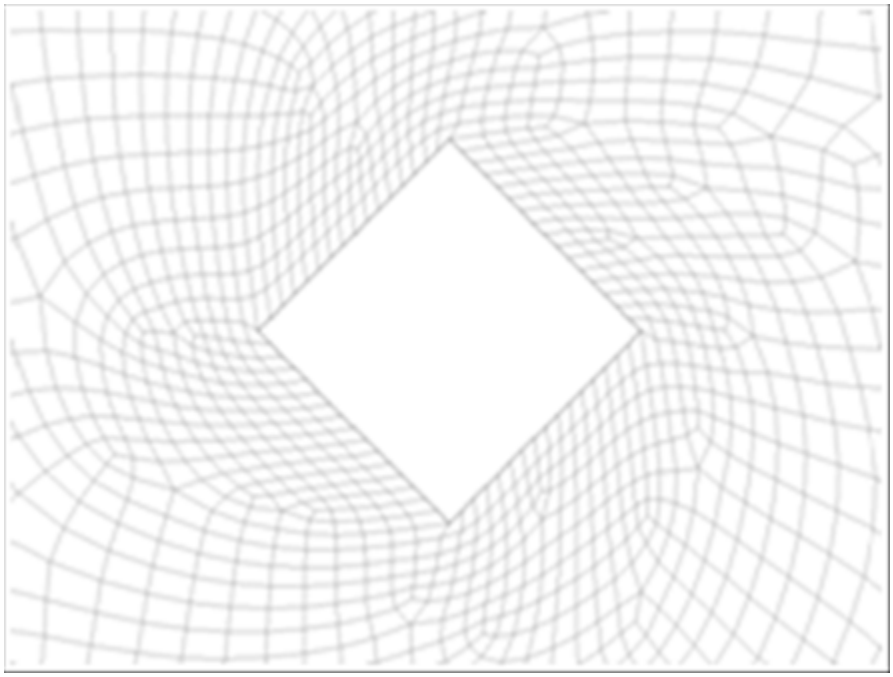
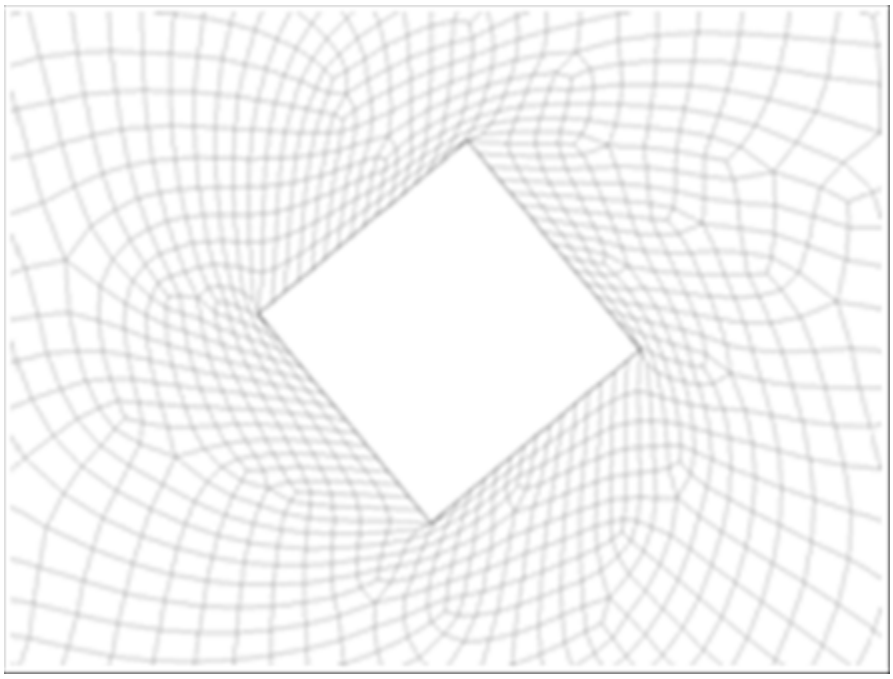


Figure 10.21: Degenerated Mesh After 40 Degree Rotation Using Spring-Based Smoothing



It should be noted that with diffusion-based smoothing the interior mesh motion is governed by the solution to Equation 10.7 (p. 650) and the prescribed boundary motion, and not by mesh irregularities. Poor quality elements or mesh defects are not smoothed out by this method, but rather move together with the pre-computed (at the begin of each mesh update) displacement velocity $\vec{u}$.

It is also worth noting that the nature of the diffusion equation is such that the resulting solution (that is, the displacement velocity $\vec{u}$) depends on the dimensionality of the problem and the type of boundary motion prescribed. To illustrate the impact of the type of boundary motion, consider the goal of boundary-distance-based diffusion (Equation 10.8 (p. 650)): to control which parts of the mesh absorb

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the boundary motion, so that you can preserve the mesh in the vicinity of the moving boundary (at the expense of the interior of the mesh). For more translational (piston-type) boundary motions, you can preserve a reasonably "thick" region of the mesh adjacent to the boundary (that is, multiple layers of cells); for rotational boundary motions, the rate of decay for the solution as you move away from the boundary is such that it can be difficult to preserve even a "thin" region. For this reason, mesh smoothing can handle translational boundary motions generally much better than rotational motions.

Although it should in most cases not be necessary, the accuracy of the solution to the diffusion equation governing the mesh motion can be controlled by selecting the maximum number of iterations, `max-iter`, and the relative residual tolerance, `relative-convergence-tolerance`, in the text interface:

`define` → `dynamic-mesh` → `controls` → `smoothing-parameters` → `max-iter`

`define` → `dynamic-mesh` → `controls` → `smoothing-parameters` → `relative-convergence-tolerance`

You can enable printing of smoothing residuals in the text interface with the command:

```
/define/dynamic-mesh/controls/smoothing-parameters/verbosity 1
```

For simulations with periodic or quasi-periodic motion, you can improve the mesh quality consistency by specifying that the diffusion-based smoothing uses a reference position, as described in Smoothing from a Reference Position (p. 657).

10.6.2.1.2.1. Diffusivity Based on Boundary Distance

Using boundary-distance-based diffusion (Equation 10.8 (p. 650)) allows you to control how the boundary motion diffuses into the interior of the domain as a function of boundary distance. Decreasing the diffusivity away from the moving boundary causes those regions to absorb more of the mesh motion, and better preserves the mesh quality near the moving boundary. This is particularly helpful for a moving boundary that has pronounced geometrical features (such as sharp corners) along with a prescribed motion that is predominantly rotational.

You can manipulate the diffusion coefficient γ (in Equation 10.8 (p. 650)) primarily by adjusting the **Diffusion Parameter** α. A range of 0 to 2 has been shown to be of practical use. A value of 0 (the default value) specifies that $\gamma=1$ and yields a uniform diffusion of the boundary motion throughout the mesh. Higher values of α preserve larger regions of the mesh near the moving boundary, and cause the regions away from the moving boundary to absorb more of the motion.

The following two figures illustrate the effect of the **Diffusion Parameter** α on the resulting mesh for a translational (piston-type) boundary motion, when the diffusivity is based on the boundary distance. In this example, an initially uniformly meshed square domain is deformed by moving the left boundary to the right.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.22: Effect of Diffusion Parameter of 0 on Interior Node Motion

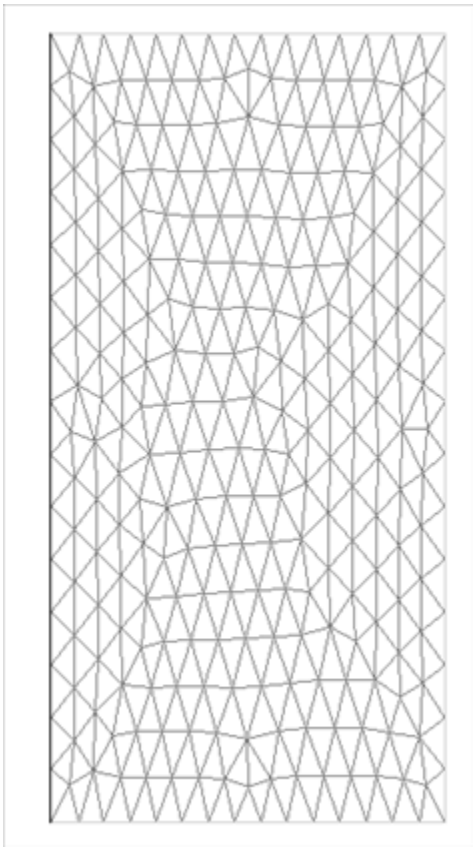
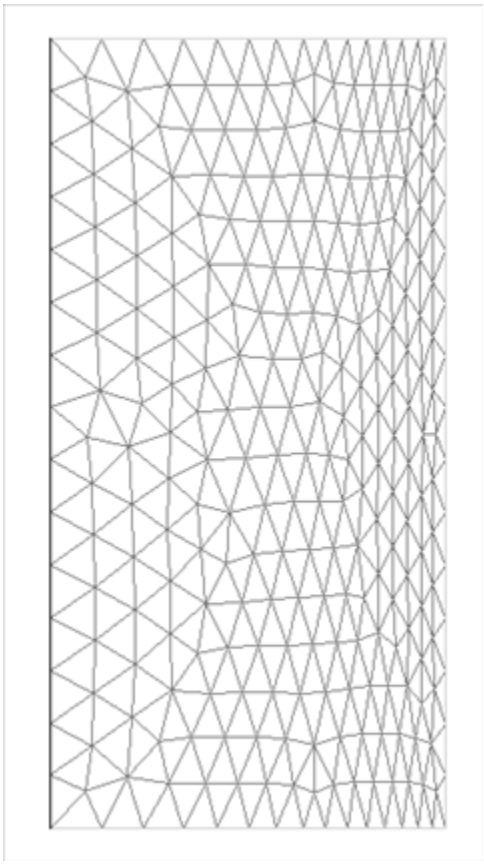


Figure 10.23: Effect of Diffusion Parameter of 1 on Interior Node Motion



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For rotational boundary motions, a value of 1.5 for the **Diffusion Parameter** α is recommended as a good starting point.

Two different methods are available for the evaluation of the boundary distance d if boundary-distance-based diffusion is used. By default, Fluent uses the "standard" boundary distance in Equation 10.8 (p. 650), which is the normalized distance to the nearest wall boundary; note that none of the other boundary types (for example, inlets, outlets, symmetry, and periodic boundaries) are considered. This method is the same as that which is used to evaluate the boundary distance for turbulence models. An example of this method is shown in Figure 10.23: Effect of Diffusion Parameter of 1 on Interior Node Motion (p. 654), where only the left and right boundaries are walls. You have the option of using a "generalized" boundary distance instead, which is the normalized distance to the nearest boundary that is not declared as deforming, regardless of type. Both methods use the largest distance found in all deforming cell zones to normalize the value.

You can specify that the generalized boundary distance is used via the following text command:

`define` → `dynamic-mesh` → `controls` → `smoothing-parameters` → `boundary-distance-method`

If the generalized boundary distance is used, an additional scalar equation for the boundary distance d will be solved as part of the solution of Equation 10.7 (p. 650).

10.6.2.1.2.2. Diffusivity Based on Cell Volume

Using cell-volume-based diffusion (Equation 10.9 (p. 650)) allows you to control how the boundary motion diffuses into the interior of the domain as a function of cell size. Decreasing the diffusivity in larger cells causes those cells to absorb more of mesh motion and therefore better preserves the cell quality of smaller cells.

You can manipulate the diffusion coefficient γ (in Equation 10.8 (p. 650)) by adjusting the **Diffusion Parameter** α. A value of 0 (the default value) specifies that $\gamma=1$ and yields a uniform diffusion of the boundary motion throughout the mesh. Higher values of α result in larger cells absorbing more of the motion than smaller cells.

Note that the cell volume used in Equation 10.9 (p. 650) is the local cell volume, normalized by the average cell volume of all deforming cell zones.

10.6.2.1.2.3. Applicability of the Diffusion-Based Smoothing Method

Diffusion-based mesh smoothing is an alternative method to spring-based smoothing. It is available for all element types, and you can use it to update any cell zone whose boundaries are moving or deforming.

Diffusion-based smoothing is computationally more expensive than spring-based smoothing, but likely results in better mesh quality (especially for non-tetrahedral / non-triangular cell zones, and for poly-hedral cells in particular) and generally allows for larger boundary deformations before breaking down.

Similar to spring-based smoothing, diffusion-based mesh smoothing can handle translational boundary deformations much better than rotational motions.

Diffusion-based smoothing is not compatible with the boundary layer smoothing method or the face region remeshing method. For more information about these methods, see Boundary Layer Smoothing Method (p. 658) and Face Region Remeshing Method (p. 676).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.6.2.1.3. Linearly Elastic Solid Based Smoothing Method

With mesh smoothing based on the linearly elastic solid model, the mesh motion is governed by the following set of equations.

$$\begin{aligned}\nabla\cdot\boldsymbol{\sigma}\left(\vec{y}\right)&=\mathbf{0}\\ \boldsymbol{\sigma}\left(\vec{y}\right)&=\lambda\left(tr\boldsymbol{\varepsilon}\left(\vec{y}\right)\right)\mathbf{I}+2\mu\boldsymbol{\varepsilon}\left(\vec{y}\right)\\ \boldsymbol{\varepsilon}\left(\vec{y}\right)&=\tfrac{1}{2}\left(\nabla\vec{y}+\left(\nabla\vec{y}\right)^{T}\right)\end{aligned} \tag{10.11}$$

where $\boldsymbol{\sigma}$ is the stress tensor, $\boldsymbol{\varepsilon}$ is the strain tensor, and $\vec{y}$ is the mesh displacement. For the solution of Equation 10.11 (p. 656) only the ratio between the shear modulus, μ, and Lamé's first parameter, λ, matters. This ratio is parameterized through Poisson's ratio:

$$\nu=\frac{1}{2\left(1+{}_{\mu/\lambda}\right)} \tag{10.12}$$

which is made available as a user input.

The boundary conditions for Equation 10.11 (p. 656) are obtained from the user-prescribed or computed (in the case of six DOF motion) boundary deformations. These imposed deformations are transferred into the interior of the deforming mesh as if the mesh was a linearly elastic solid with the given material properties. On deforming boundaries you can either specify a geometry along which the mesh can slide, or leave the geometry unspecified. If a geometry is specified for the deforming boundary, then the boundary conditions are such that the deformation normal to the boundary vanishes and the stress tangential to the boundary is zero. If the geometry of the deforming boundary is unspecified, then the deforming boundary can also deform in the normal direction and the boundary conditions are such that the traction is zero in all directions. See Deforming Motion (p. 731) for details of how to specify geometry on deforming zones.

The linear system in Equation 10.11 (p. 656) is solved using a finite element discretization and the mesh displacements for the interior and deforming boundary nodes are obtained directly at the nodes. The accuracy to which the linear system is solved can be controlled by the maximum number of iterations `max-iter`, and the relative residual tolerance `relative-convergence-tolerance` in the text interface:

`define` → `dynamic-mesh` → `controls` → `smoothing-parameters` → `max-iter`

`define` → `dynamic-mesh` → `controls` → `smoothing-parameters` → `relative-convergence-tolerance`

The linearly elastic solid mesh smoothing model supports constant material properties only. The permissible range for Poisson's ratio, ν, is between -1.0 and 0.5.

For simulations with periodic or quasi-periodic motion, you can improve the mesh quality consistency by specifying that the linearly elastic solid mesh smoothing uses a reference position, as described in Smoothing from a Reference Position (p. 657).

10.6.2.1.3.1. Applicability of the Linearly Elastic Solid Based Smoothing Method

Mesh smoothing based on the linearly elastic solid model is an alternative method to diffusion-based and spring-based smoothing. Most of the properties and limitations discussed for diffusion-based smoothing (Applicability of the Diffusion-Based Smoothing Method (p. 655)) also apply to the linearly elastic solid model, particularly the mesh quality degradation for rotational motions. The linearly elastic

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

solid model is computationally more expensive than diffusion-based smoothing, but for some meshes and mesh motions preserves the mesh quality better.

The current implementation with constant material properties can be a limitation compared with diffusion-based smoothing with non-uniform diffusivity. In cases with rotational boundary motion and sharp corners it may be advantageous to use diffusion-based smoothing with boundary-distance-dependent diffusivity.

The option to leave the geometry of deforming face zones unspecified is only available with the linearly elastic solid smoothing method.

The linearly elastic solid smoothing method supports triangular and quadrilateral elements in 2D and tetrahedral, hexahedral, wedge, and pyramid cells in 3D. It cannot be applied if the deforming cell zone contains polyhedral cells or hanging nodes. In such cases diffusion-based smoothing is recommended.

Linearly elastic solid smoothing is not compatible with the boundary layer smoothing method or the face region remeshing method. For more information about these methods, see Boundary Layer Smoothing Method (p. 658) and Face Region Remeshing Method (p. 676).

10.6.2.1.4. Smoothing from a Reference Position

When the mesh smoothing is based on diffusion or the linearly elastic solid model, you have the option of specifying that the smoothing uses a reference position. This can be helpful when performing many cycles of periodic or quasi-periodic motion: when you ensure that the smoothing is always done from the same reference position, the mesh quality may remain more consistent from cycle to cycle. Note that no mesh manipulation should be done to the zone undergoing such smoothing.

To enable this option, use the following text command:

`define` → `dynamic-mesh` → `controls` → `smoothing-parameters` → `smooth-from-reference-position?`

The reference position is saved the first time you perform smoothing after enabling this option. The reference position is not saved to the case or data file, so restarting from a case file will imply that a different reference position is used after restart. This still provides the benefit of consistent quality, but the resulting meshes might be slightly different from a continuous run. The impact of such slight mesh changes on the solution results will be small, except for cases with a high degree of mesh dependence; for such exceptional cases, it is recommended that you evaluate the original mesh to ensure that it is suitable for the problem.

10.6.2.1.5. Laplacian Smoothing Method

Laplacian smoothing is the most commonly used and the simplest mesh smoothing method. This method adjusts the location of each mesh vertex to the geometric center of its neighboring vertices. This method is computationally inexpensive but it does not guarantee an improvement on mesh quality, since repositioning a vertex by Laplacian smoothing can result in poor quality elements. To overcome this problem, ANSYS Fluent only relocates the vertex to the geometric center of its neighboring vertices if and only if there is an improvement in the mesh quality (that is, the skewness has been improved).

This improved Laplacian smoothing can be enabled on deforming boundaries only (that is, the zone with triangular elements in 3D and zones with linear elements in 2D). The computation of the node positions works as follows:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\overline{\vec{x}}_i^m = \frac{\sum_j^{n_i} \vec{x}_i^m}{n_i} \tag{10.13}$$

where $\overline{\vec{x}}_i^m$ is the averaged node position of node i at iteration m, $\vec{x}_j^m$ is the node position of neighbor node of $\vec{x}_i^m$ at iteration m, and n_i is the number nodes neighboring node i. The new node position $\vec{x}_i^{m+1}$ is then computed as follows:

$$\vec{x}_i^{m+1} = \vec{x}_i^m (1-\beta) + \overline{\vec{x}}_i^m \beta \tag{10.14}$$

where β is the Laplace node relaxation factor.

This update only happens if the maximum skewness of all faces adjacent to $\vec{x}_i^{m+1}$ is improved in comparison to $\vec{x}_i^m$.

For details on applying Laplacian smoothing to either a cell zone (with 2.5D remeshing) or a face zone, see Smoothing Methods (p. 644) or Deforming Motion (p. 731), respectively.

10.6.2.1.6. Boundary Layer Smoothing Method

The boundary layer smoothing method is used to deform the boundary layer mesh during a moving-deforming mesh simulation. For cases that apply mesh motion (either **Rigid Body** or **User-Defined**) to a face zone with adjacent boundary layers, the boundary layers can be made to deform accordingly by enabling **Deform Adjacent Boundary Layer with Zone** for the face zone in the Dynamic Mesh Zones Dialog Box (p. 2490). With boundary layer smoothing enabled, the nodal coordinates of each cell in the boundary layer zone are updated with the same displacement vector as the corresponding nodes on the underlying face zone. The boundary layer smoothing method can be applied to boundary layer zones of all mesh types (that is, wedges and hexahedra in 3D, quadrilaterals in 2D).

Consider the example below, where a UDF of the form `DEFINE_GRID_MOTION` provides the moving-deforming mesh model with the locations of the nodes located on the compliant strip on an idealized airfoil. The node motion varies sinusoidally in time and space (compare Figure 10.24: The Undeformed Mesh (p. 659) with Figure 10.25: The Deformed Mesh (p. 659)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.24: The Undeformed Mesh

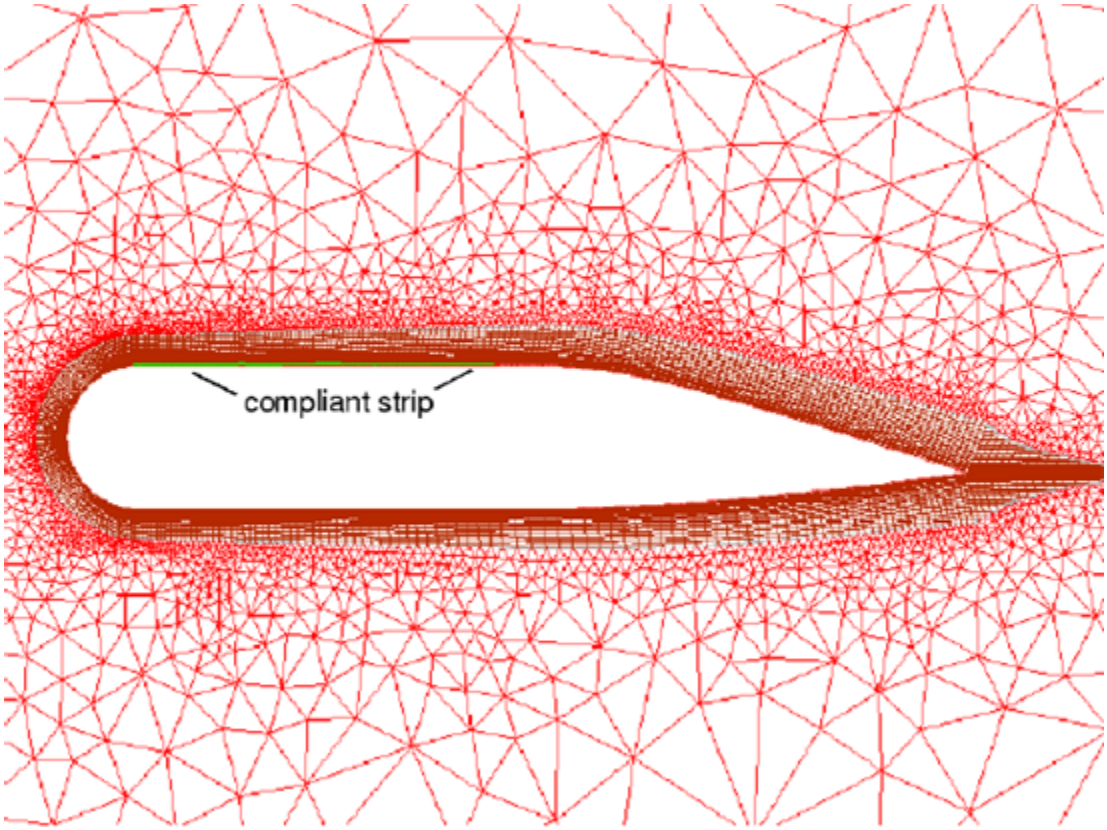
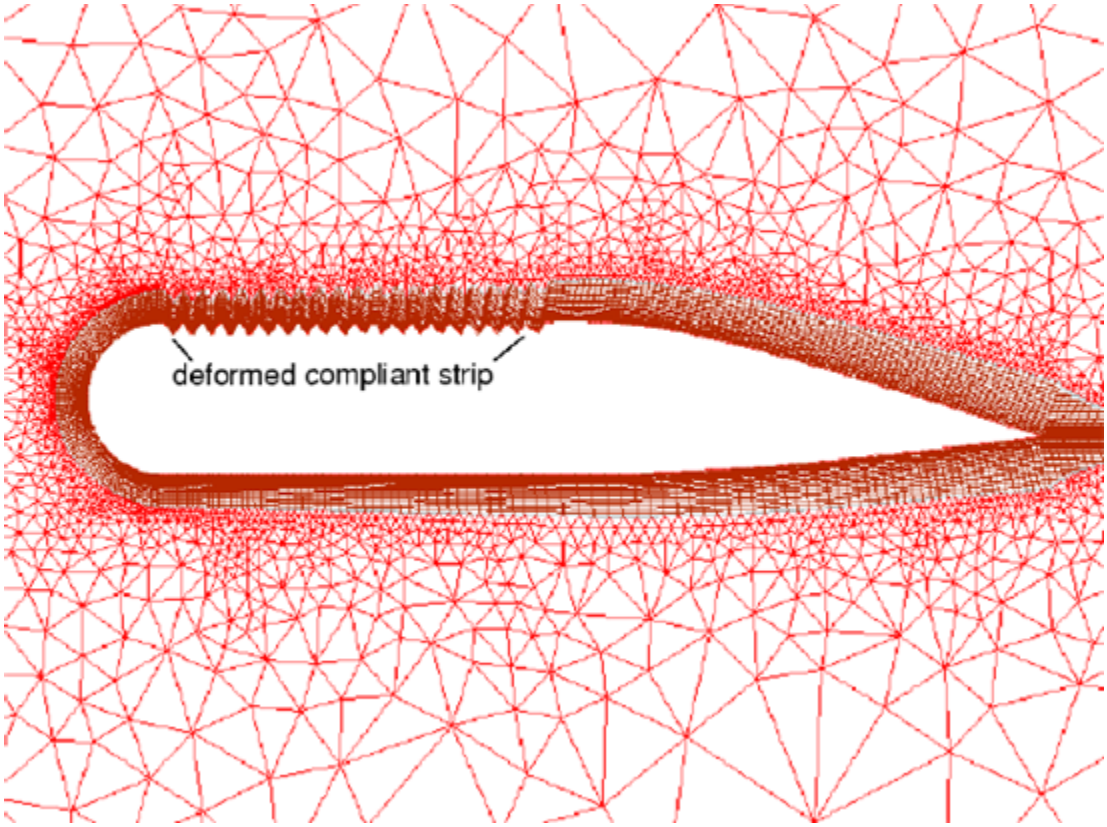


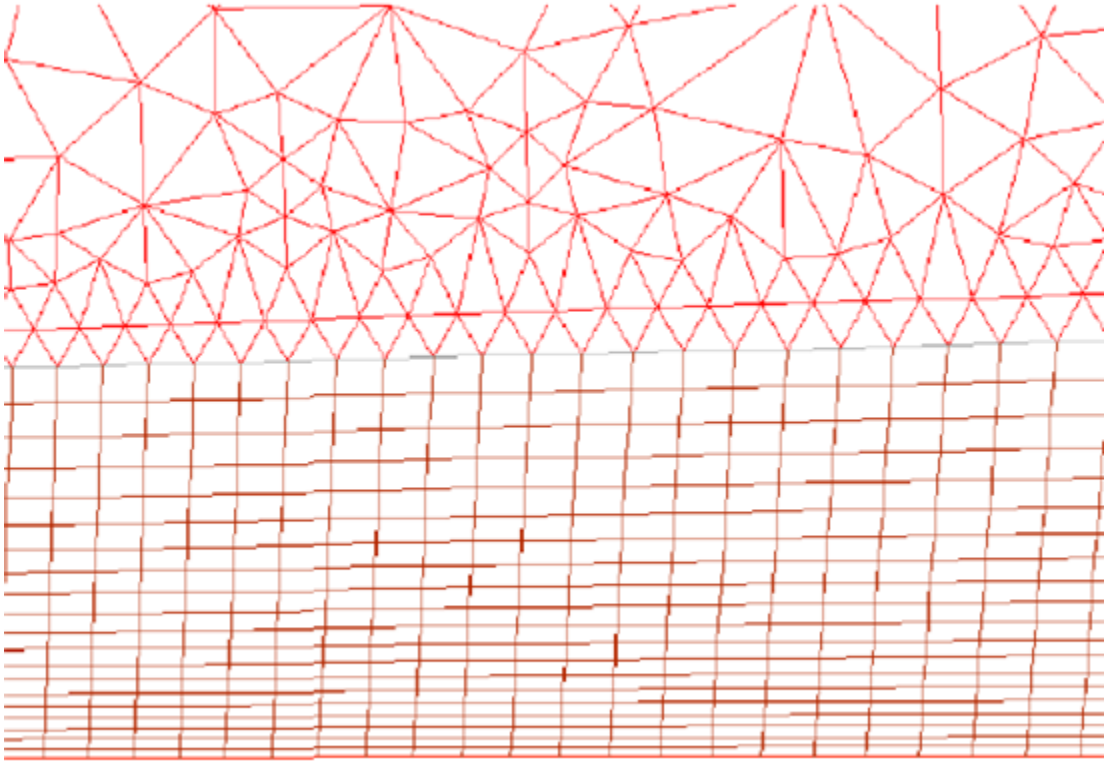
Figure 10.25: The Deformed Mesh



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

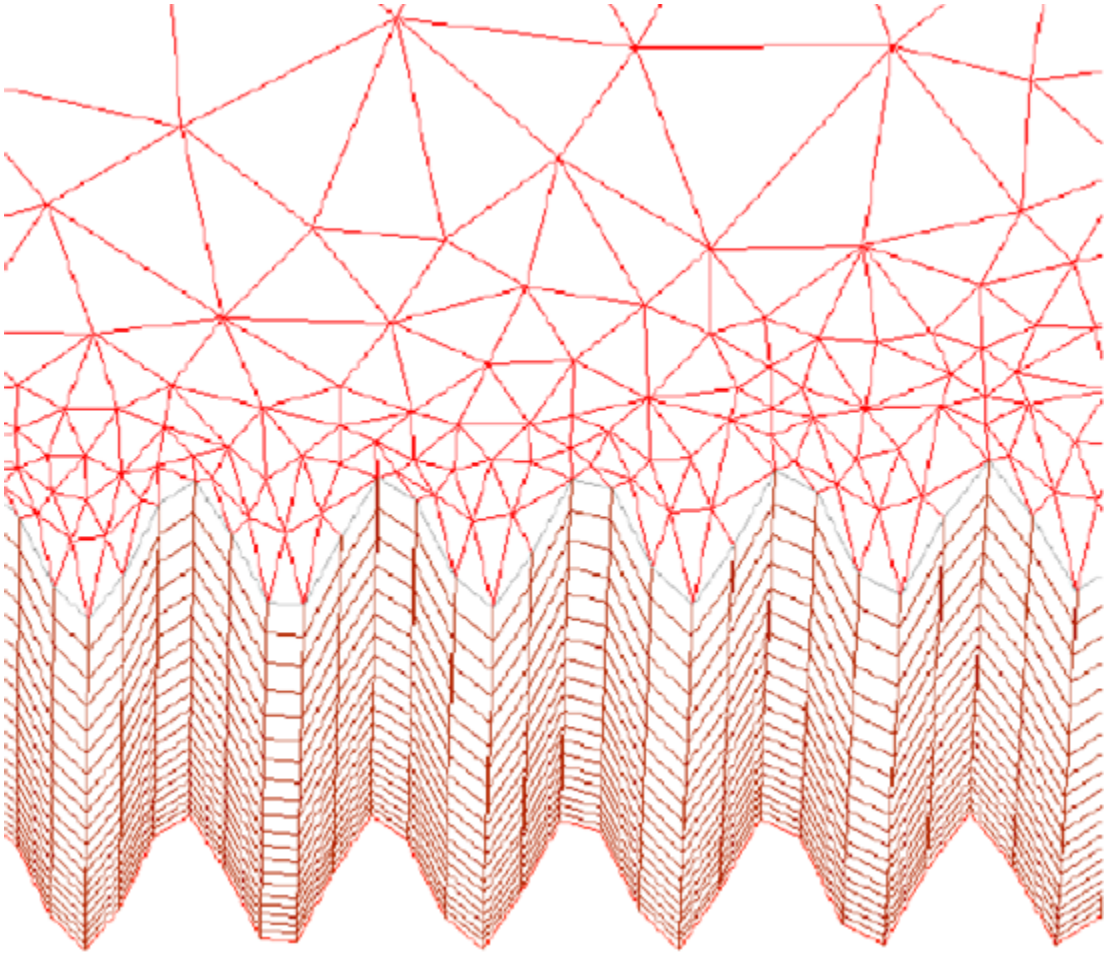
As a result of the boundary layer smoothing, the cells adjacent to the deforming wall are also deformed in order to preserve the quality of boundary layer zone. If you compare Figure 10.26: Zooming into the Undeformed Compliant Strip (p. 660) with Figure 10.27: Zooming into the Deformed Compliant Strip with Boundary Layer Smoothing Applied (p. 661), you can see how the boundary layer cells have been deformed according to the motion of the nodes on the compliant strip.

Figure 10.26: Zooming into the Undeformed Compliant Strip



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.27: Zooming into the Deformed Compliant Strip with Boundary Layer Smoothing Applied



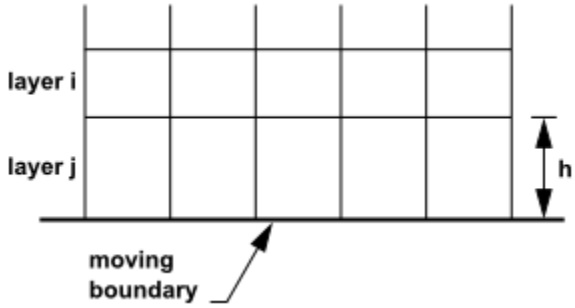
Typically, the boundary layer smoothing method preserves the height of the boundary layer cells adjacent to the deformed face zone. However, note that this approach is primarily intended for translational motion. If the faces undergo substantial rotation, the boundary layer cells may become skewed. See Smoothing Methods (p. 644) and Specifying Boundary Layer Deformation Smoothing (p. 737) for details about enabling smoothing and defining a moving and deforming boundary layer, respectively. Note that boundary layer smoothing is compatible with spring-based smoothing only. It cannot be used with diffusion-based smoothing or linearly elastic solid smoothing. Also note that the boundary layer smoothing method will work whether or not you have segregated the boundary layer elements into a separate cell zone.

10.6.2.2. Dynamic Layering

In prismatic (hexahedral and/or wedge) mesh zones, you can use dynamic layering to add or remove layers of cells adjacent to a moving boundary, based on the height of the layer adjacent to the moving surface. The dynamic mesh model in ANSYS Fluent allows you to specify an ideal layer height on each moving boundary. The layer of cells adjacent to the moving boundary (layer j in Figure 10.28: Dynamic Layering (p. 662)) is split or merged with the layer of cells next to it (layer i in Figure 10.28: Dynamic Layering (p. 662)) based on the height (h) of the cells in layer j.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.28: Dynamic Layering



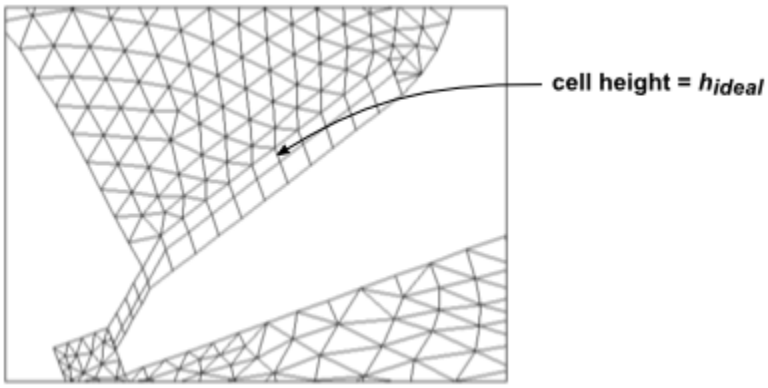
If the cells in layer j are expanding, the cell heights are allowed to increase until

$$h_{min} > (1+\alpha_s)h_{ideal} \tag{10.15}$$

where h_{min} is the minimum cell height of cell layer j, h_{ideal} is the ideal cell height, and α_s is the layer split factor. Note that ANSYS Fluent allows you to define h_{ideal} as either a constant value or a value that varies as a function of time or crank angle. When the condition in Equation 10.15 (p. 662) is met, the cells are split based on the specified layering option. This option can be height based or ratio based.

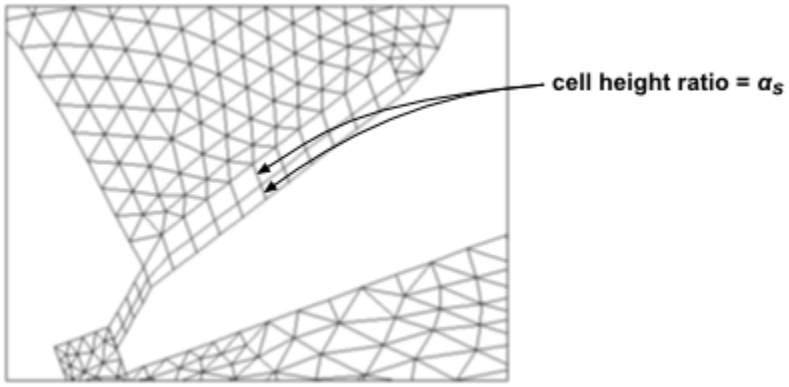
With the height-based option, the cells are split to create a layer of cells with constant height h_{ideal} and a layer of cells of height $h-h_{ideal}$. With the ratio-based option the cells are split such that, locally, the ratio of the new cell heights to old cell heights is exactly α_s everywhere. Figure 10.29: Results of Splitting Layer with the Height-Based Option (p. 662) and Figure 10.30: Results of Splitting Layer with the Ratio-Based Option (p. 663) show the result of splitting a layer of cells above a valve geometry using the height-based and ratio-based option.

Figure 10.29: Results of Splitting Layer with the Height-Based Option



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.30: Results of Splitting Layer with the Ratio-Based Option



If the cells in layer j are being compressed, they can be compressed until

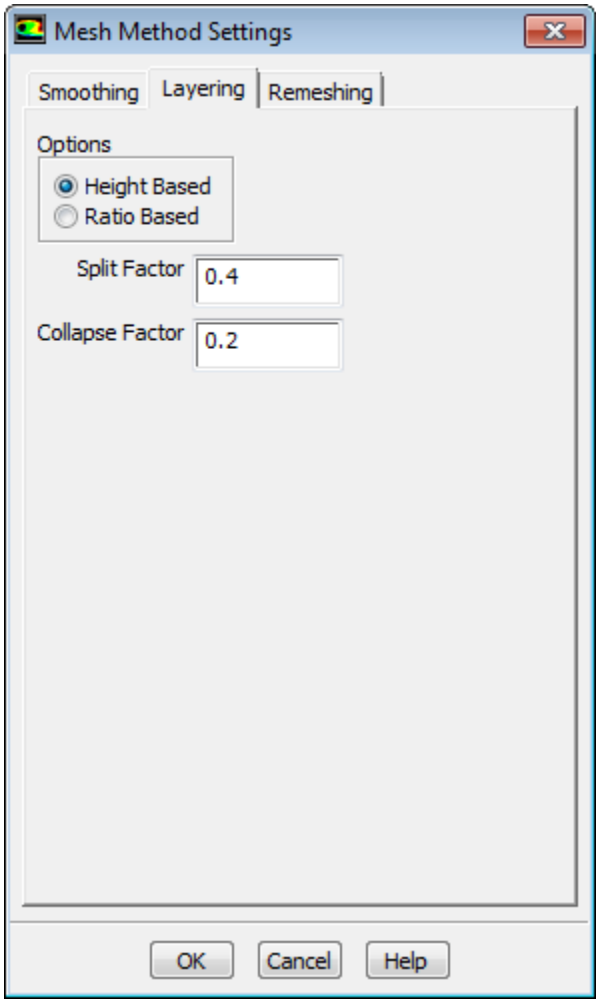
$$h_{min} < \alpha_c h_{ideal} \tag{10.16}$$

where α_c is the layer collapse factor. When this condition is met, the compressed layer of cells is merged into the layer of cells above the compressed layer in Figure 10.28: Dynamic Layering (p. 662); that is, the cells in layer j are merged with those in layer i.

To enable dynamic layering, enable the **Layering** option under **Mesh Methods** in the Dynamic Mesh Task Page (p. 2471) (Figure 10.31: The Layering Tab in the Mesh Method Settings Dialog Box (p. 664)). The layering control is specified in the **Layering** tab, which can be displayed by clicking **Settings...**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.31: The Layering Tab in the Mesh Method Settings Dialog Box



You can control how a cell layer is split by specifying either **Height Based** or **Ratio Based** under **Options**. Note that for **Height Based**, the height of the cells in a particular new layer will be constant, but you can choose to have this height vary from layer to layer as a function of time or crank angle when you specify the **Cell Height** in the **Dynamic Mesh Zones** dialog box (see Specifying the Motion of Dynamic Zones (p. 722) for further details).

The **Split Factor** and **Collapse Factor** (α_s in Equation 10.15 (p. 662) and α_c in Equation 10.16 (p. 663), respectively) are the factors that determine when a layer of cells (hexahedra or wedges in 3D, or quadrilaterals in 2D) that is next to a moving boundary is split or merged with the adjacent cell layer, respectively.

10.6.2.2.1. Applicability of the Dynamic Layering Method

You can use the dynamic layering method to split or merge cells adjacent to any moving boundary provided the following conditions are met:

- All cells adjacent to the moving face zone are either wedges or hexahedra (quadrilaterals in 2D) even though the cell zone may contain mixed cell shapes.
- The cell layers must be completely bounded by one-sided face zones, except when sliding interfaces are used (see Applicability of the Face Region Remeshing Method (p. 680)).

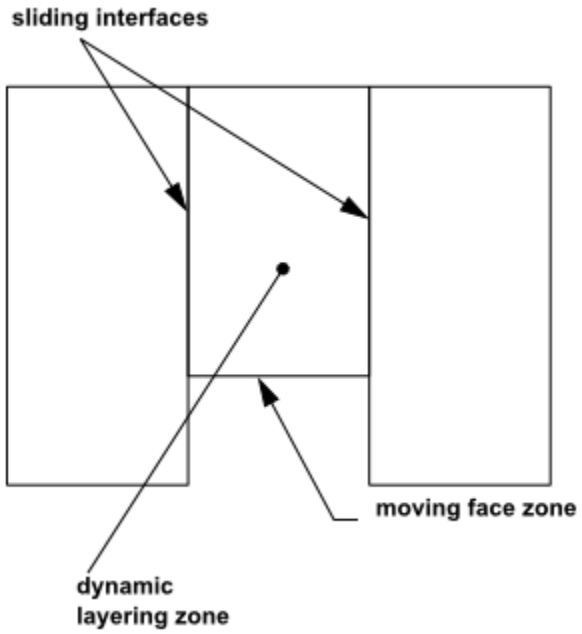
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Note that you cannot use the dynamic layering method in conjunction with hanging node adaption. For more information on hanging node adaption, see Hanging Node Adaption in the Theory Guide.

If the moving boundary is an internal zone, cells on both sides (possibly with different ideal cell layer heights) of the internal zone are considered for dynamic layering.

If you want to use dynamic layering on cells adjacent to a moving wall that do not span from boundary to boundary, you must separate those cells that are involved in the dynamic layering and use the sliding interfaces capability in ANSYS Fluent to transition from the deforming cells to the adjacent non-deforming cells (see Figure 10.32: Use of Sliding Interfaces to Transition Between Adjacent Cell Zones and the Dynamic Layering Cell Zone (p. 665)). For a moving interior face, the zones must be separated such that they are either expanding or collapsing on the same side. No one zone can consist of both expanding and collapsing layers.

Figure 10.32: Use of Sliding Interfaces to Transition Between Adjacent Cell Zones and the Dynamic Layering Cell Zone



10.6.2.3. Remeshing Methods

On zones with a triangular or tetrahedral mesh, the spring-based smoothing method (described in Spring-Based Smoothing (p. 646)) is normally used. When the boundary displacement is large compared to the local cell sizes, the cell quality can deteriorate or the cells can become degenerate. This will invalidate the mesh (for example, result in negative cell volumes) and consequently, will lead to convergence problems when the solution is updated to the next time step.

To circumvent this problem, ANSYS Fluent agglomerates cells that violate the skewness or size criteria and locally remeshes the agglomerated cells or faces. If the new cells or faces satisfy the skewness criterion, the mesh is locally updated with the new cells (with the solution interpolated from the old cells). Otherwise, the new cells are discarded and the old cells are retained.

ANSYS Fluent includes several remeshing methods that include local cell remeshing, zone remeshing, local face remeshing (for 3D flows only), face region remeshing, CutCell zone remeshing (for 3D flows only), and 2.5D surface remeshing (for 3D flows only). The remeshing methods are suitable for particular kinds of cell types:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

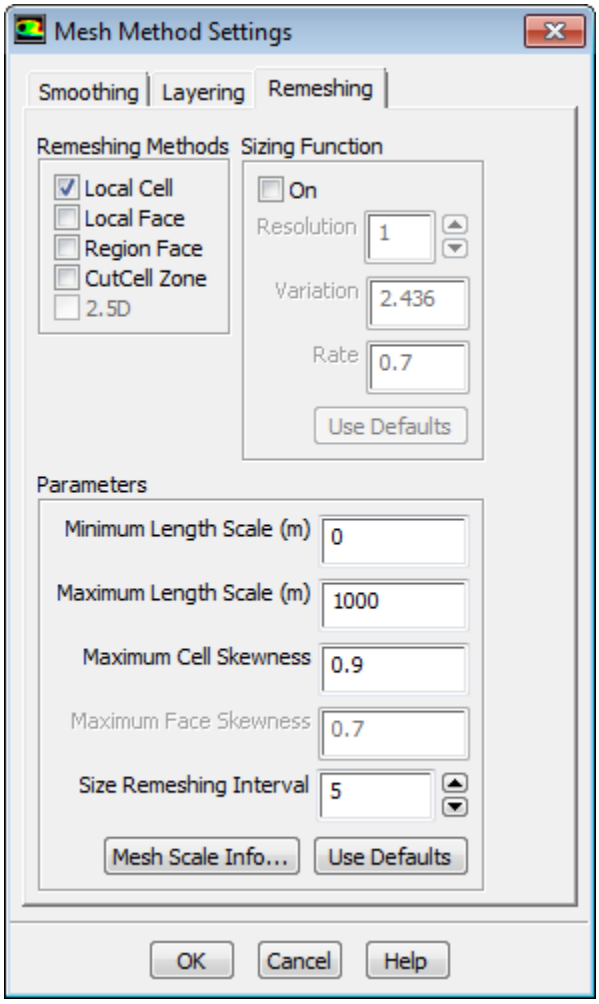
- The local cell remeshing method only affects triangular and tetrahedral cell types in the mesh (that is, in mixed cell zones the non-triangular/tetrahedral cells are skipped).
- The local face remeshing method is available in 3D only and can remesh tetrahedral cells and wedge cells in boundary layer meshes.
- The zone remeshing method replaces all cell types with triangular tetrahedral cells (in 2D and 3D domains, respectively), and can remesh and produce wedge cells in 3D boundary layer meshes.
- The face region remeshing method is applied to triangular cells in 2D, and tetrahedral cells in 3D. In 3D domains, the face region remeshing method can also remesh and produce wedge cells in 3D boundary layer meshes.
- The CutCell zone remeshing method works for all cell types.
- The 2.5D remeshing method only works on hexagonal meshes or wedge cells extruded from triangular surface elements.

To enable remeshing methods, enable the **Remeshing** option under **Mesh Methods** in the Dynamic Mesh Task Page (p. 2471) (Figure 10.13: The Dynamic Mesh Task Page (p. 643)). Click the **Settings...** button to open the **Mesh Method Settings** dialog box, where you can specify the remeshing method and parameters in the **Remeshing** tab (Figure 10.33: The Remeshing Tab in the Mesh Method Settings Dialog Box (p. 667)).

You can view the vital statistics of your mesh by clicking the **Mesh Scale Info...** button at the bottom of the Mesh Method Settings Dialog Box (p. 2474). This button opens the Mesh Scale Info Dialog Box (p. 2477), where you can view the minimum and maximum length scale values, as well as the maximum cell and face skewness values.

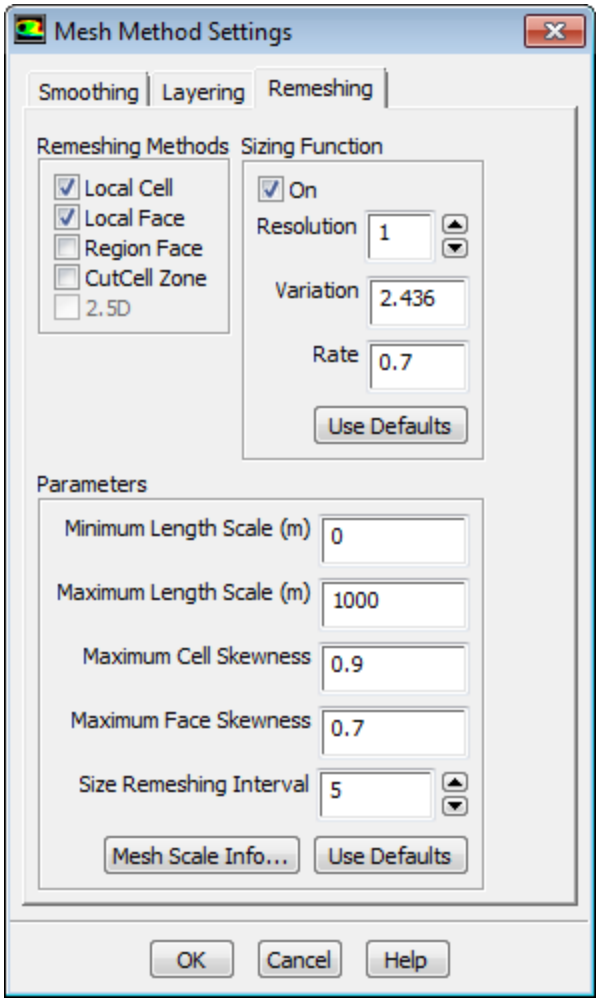
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.33: The Remeshing Tab in the Mesh Method Settings Dialog Box



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.34: The Remeshing Tab in the Mesh Method Settings Dialog Box Using the Sizing Function Option



10.6.2.3.1. Local Remeshing Method

Using the local remeshing method (that is, local cell remeshing, with or without local face remeshing), ANSYS Fluent marks cells based on cell skewness and minimum and maximum length scales as well as an optional sizing function.

ANSYS Fluent evaluates each cell and marks it for remeshing if it meets one or more of the following criteria:

- It has a skewness that is greater than a specified maximum skewness.
- It is smaller than a specified minimum length scale.
- It is larger than a specified maximum length scale.
- Its height does not meet the specified length scale (at moving face zones, for example, above a moving piston).

If local remeshing is not able to reduce the maximum cell skewness sufficiently, then the cell zone remeshing method is used to automatically remesh all of the cells in the cell zone, as well as the faces of all adjacent deforming dynamic face zones (see Cell Zone Remeshing Method (p. 676) for details). The

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

maximum allowable cell skewness is set to be 0.98 by default. The cell zone remeshing method gives the mesher more flexibility to create a new mesh of better quality than the local cell remeshing method. The automatic remeshing of cell zones can be disabled using the following text command:

`define` → `dynamic-mesh` → `controls` → `remeshing-parameter` → `zone-remeshing`

10.6.2.3.1.1. Local Cell Remeshing Method

As previously mentioned, in local cell remeshing, ANSYS Fluent agglomerates cells based on skewness, size, and height (adjacent moving face zones) prior to the movement of the boundary. The size criteria are specified with **Minimum Length Scale** and **Maximum Length Scale**. Cells with length scales below the minimum length scale and above the maximum length scale are marked for remeshing. The value of **Maximum Cell Skewness** indicates the desired skewness of the mesh. By default, the **Maximum Cell Skewness** is set to 0.9 for 3D simulations and 0.7 for 2D simulations. Cells with skewness above the maximum skewness are marked for remeshing.

The marking of cells based on skewness is done at every time step when the local remeshing method is enabled. However, marking based on size and height is performed between the specified **Size Remeshing Interval** since the change in cell size distribution is typically small over one time step.

By default, ANSYS Fluent replaces the agglomerated cells only if the quality of the remeshed cells has improved.

10.6.2.3.1.2. Local Face Remeshing Method

The local face remeshing method only applies to 3D geometries. You can apply this method to the boundaries of deforming or user-defined zone types, so that ANSYS Fluent will mark faces (and adjacent cells) based on the face skewness, and then remesh them.

Local face remeshing also allows the remeshing of wedge cells in boundary layers at those boundaries. The detection of boundary layers (as well as the wedge element height distribution and number of layers) is automatic and does not require your input.

To apply local face remeshing, perform the following steps:

1. Enable the **Local Face** option in the **Remeshing** tab of the Mesh Method Settings Dialog Box (p. 2474), and set the **Maximum Face Skewness** to an appropriate global value.

2. For each deforming or user-defined boundary zone on which you want local face remeshing applied, enable the **Local** option (under **Remeshing Methods**) in the **Meshing Options** tab of the Dynamic Mesh Zones Dialog Box (p. 2490). (Note that for deforming zones, you must first enable the **Remeshing** option under **Methods**, and then **Local** will be enabled by default.) You can also revise the **Maximum Skewness** field in this tab if you want a value that is different from the global setting. For further details on the zone setup, see Deforming Motion (p. 731) and/or User-Defined Motion (p. 735).

10.6.2.3.1.2.1. Applicability of the Local Face Remeshing Method

When you enable local face remeshing for a boundary, the faces can be remeshed only if the following conditions are met:

- The faces are triangular.

- The agglomerated faces do not span multiple zones or feature edges.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that you cannot use the local face remeshing method in conjunction with hanging node adaption. For more information on hanging node adaption, see Hanging Node Adaption in the Theory Guide.

10.6.2.3.1.3. Local Remeshing Based on Size Functions

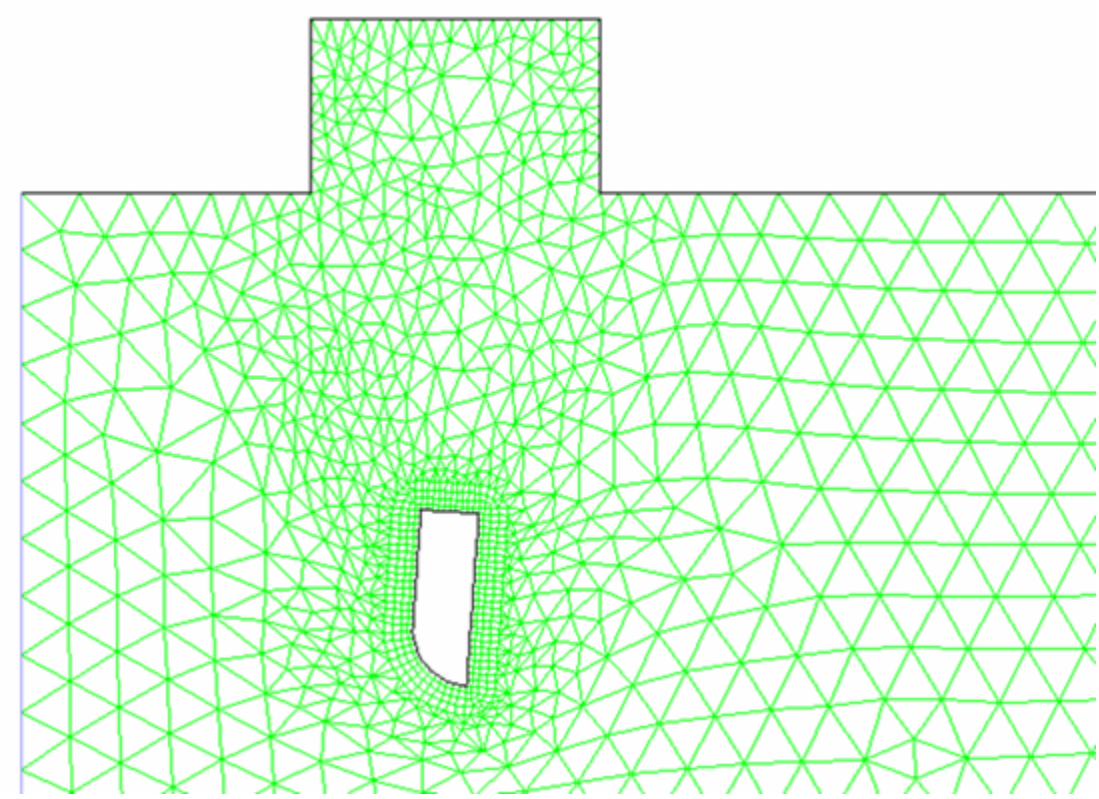
Instead of marking cells based on minimum and maximum length scales, ANSYS Fluent also marks cells based on the size distribution generated by the sizing function if the **Sizing Function** option is enabled.

Cells can be marked using size functions only with the following remeshing methods:

- local cell remeshing
- 2.5D surface remeshing (as described in 2.5D Surface Remeshing Method (p. 685))

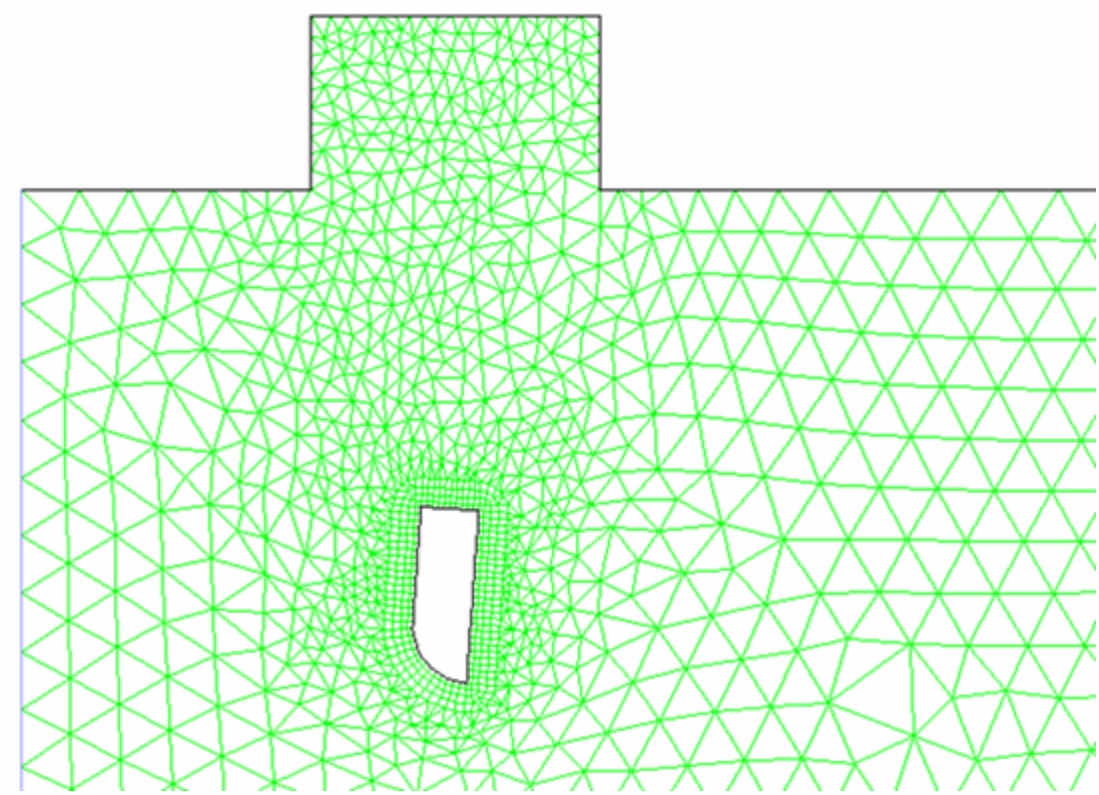
Figure 10.36: Mesh at the End of a Dynamic Mesh Simulation With Size Functions (p. 671) demonstrates the advantages of using size functions for local remeshing.

Figure 10.35: Mesh at the End of a Dynamic Mesh Simulation Without Size Functions



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.36: Mesh at the End of a Dynamic Mesh Simulation With Size Functions

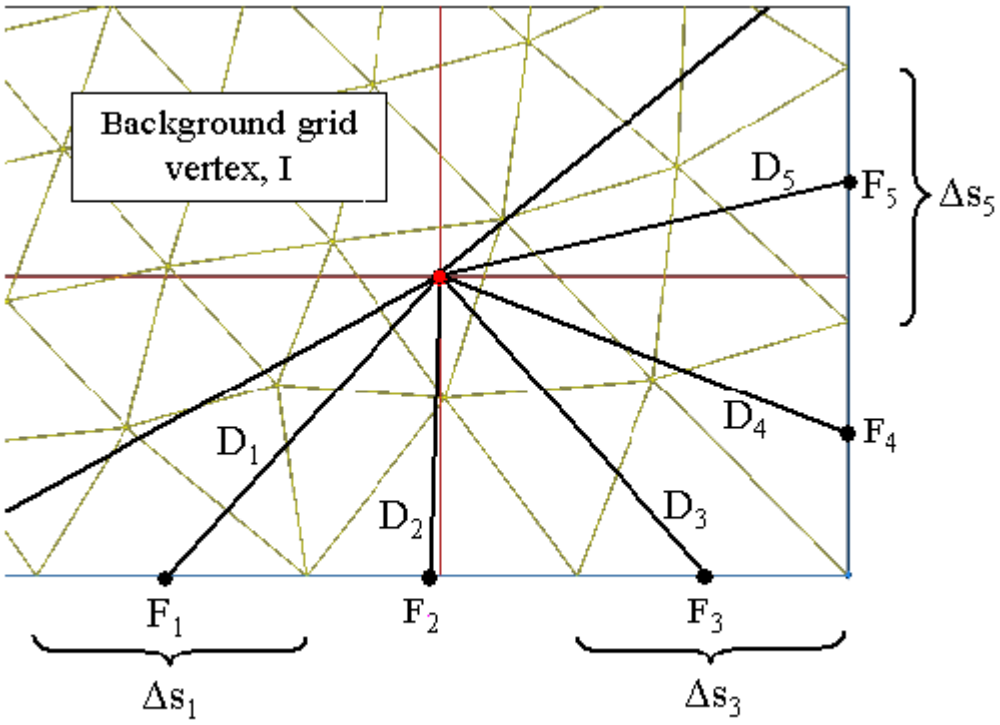


In determining the sizing function, ANSYS Fluent draws a bounding box around the zone that is approximately twice the size of the zone, and locates the shortest feature length within each fluid zone. ANSYS Fluent then subdivides the bounding box based on the shortest feature length and the **Size Function Resolution** that you specify. This allows ANSYS Fluent to create a background mesh.

You control the resolution of the background mesh and a background mesh is created for each fluid zone. The shortest feature length is determined by shrinking a second box around the object, and then selecting the shortest edge on that box. The size function is evaluated at the vertex of each individual background mesh.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.37: Size Function Determination at Background Mesh Vertex I



As seen in Figure 10.37: Size Function Determination at Background Mesh Vertex I (p. 672), the local value of the size function SF_I is defined by

$$SF_I = \left(\frac{\Sigma \frac{1}{D_J} \Delta s_J}{\Sigma \frac{1}{D_J}} \right) \tag{10.17}$$

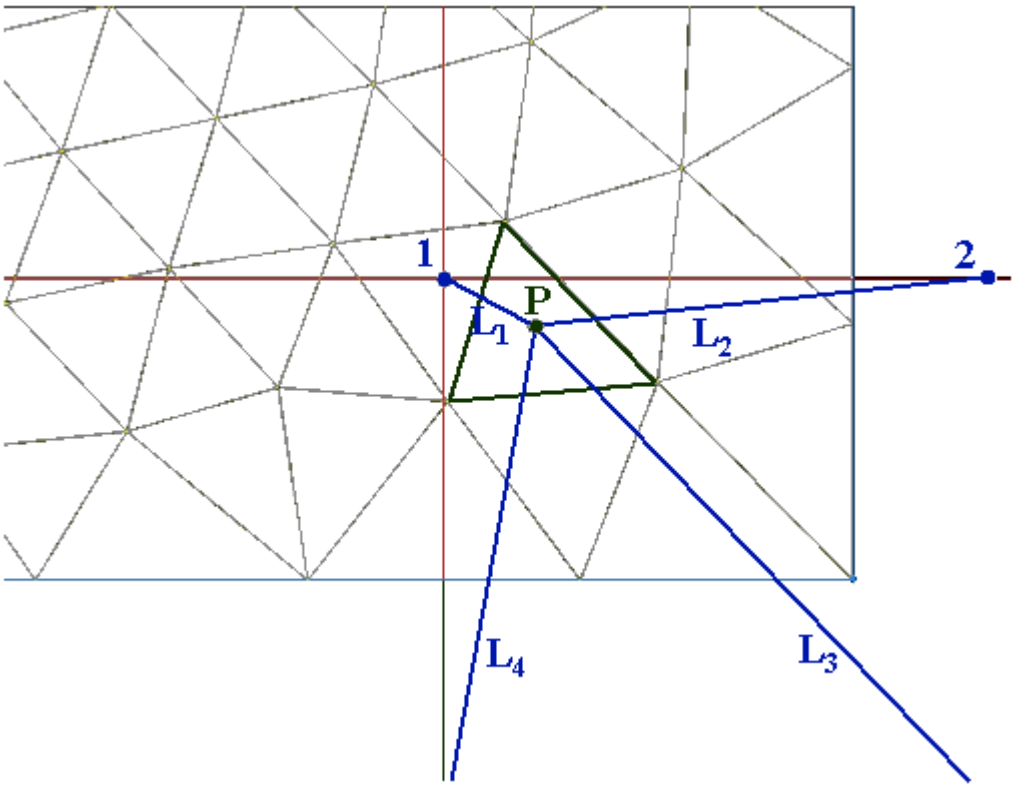
where D_J is the distance from vertex I on the background mesh to the centroid of boundary cell J and Δs_J is the mesh size (length) of boundary cell J.

The size function is then smoothed using Laplacian smoothing. ANSYS Fluent then interpolates the value of the size function by calculating the distance L_I from a given cell centroid P to the background mesh vertices that surround the cell (see Figure 10.38: Interpolating the Value of the Size Function (p. 673)). The intermediate value of the size function $size_b$ at the centroid is computed from

$$size_b = \left(\frac{\Sigma SF_I \frac{1}{L_I}}{\Sigma \frac{1}{L_I}} \right) \tag{10.18}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

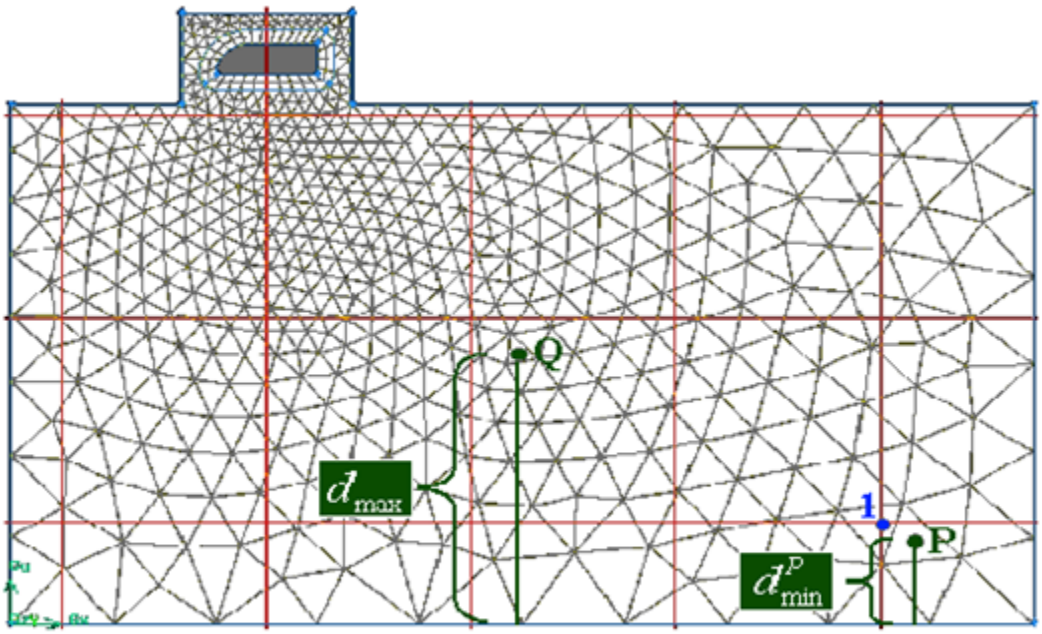
Figure 10.38: Interpolating the Value of the Size Function



Next, a single point Q is located within the domain (see Figure 10.39: Determining the Normalized Distance (p. 674)) that has the largest distance d_{max} to the nearest boundary to it. The normalized distance d_b for the given centroid P is given by

$$d_b = \frac{d^P_{min}}{d_{max}} \tag{10.19}$$

Figure 10.39: Determining the Normalized Distance



Using the parameters α and β (the **Size Function Variation** and the **Size Function Rate**, respectively), you can write the final value $size_P$ of the size function at point P as

$$size_P = size_b \times \left(1 + \alpha \times d_b^{1+2\beta}\right) = size_b \times \gamma \tag{10.20}$$

where $size_b$ is the intermediate value of the size function at the cell centroid.

Note that α is the size function *variation*. Positive values mean that the cell size increases as you move away from the boundary. Since the maximum value of d_b is one, the maximum cell size becomes

$$size_{P,max} = size_b \times \left(1 + \alpha\right) = size_b \times \gamma_{max} \tag{10.21}$$

therefore, α is really a measure of the maximum cell size.

The factor γ is computed from

$$\gamma = 1 + \alpha d_b^{1+2\beta} \quad \text{if } \alpha > 0 \tag{10.22}$$

$$\gamma = 1 + \alpha d_b^{\frac{1}{1-\beta}} \quad \text{if } \alpha < 0 \tag{10.23}$$

You can use **Size Function Variation** (or α) to control how large or small an interior cell can be with respect to its closest boundary cell. α ranges from -1 to ∞, an α value of 0.5 indicates that the interior cell size can be, at most, 1.5 the size of the closest boundary cell. Conversely, an α value of -0.5 indicates that the cell size interior of the boundary can be half of that at the closest boundary cell. A value of 0 indicates a constant size distribution away from the boundary.

You can use **Size Function Rate** (or β) to control how rapidly the cell size varies from the boundary. The value of β should be specified such that $-0.99 < \beta < +0.99$. A positive value indicates a slower transition from the boundary to the specified **Size Function Variation** value. Conversely, a negative

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

value indicates a faster transition from the boundary to the **Size Function Variation** value. A value of 0 indicates a linear variation of cell size away from the boundary.

You can also control the resolution of the sizing function with **Size Function Resolution**. The resolution determines the size of the background bins used to evaluate the size distribution with respect to the shortest feature length of the current mesh. By default, the **Size Function Resolution** is 3 in 2D problems, and 1 in 3D problems.

A set of default values (based on the current mesh) is automatically generated if you click **Use Defaults**.

In summary, the sizing function is a distance-weighted average of all mesh sizes on all boundary faces (both stationary and moving boundaries). The sizing function is based on the sizes of the boundary cells, with the size computed from the cell volume by assuming a perfect (equilateral) triangle in 2D and a perfect tetrahedron in 3D. You can control the size distribution by specifying the **Size Function Variation** and the **Size Function Rate**. If you have enabled the **Sizing Function** option, ANSYS Fluent will agglomerate a cell if

$$size \notin \left[\frac{4}{5}\gamma size_b, \frac{5}{4}\gamma size_b\right] \tag{10.24}$$

where γ is a factor defined by Equation 10.22 (p. 674) and Equation 10.23 (p. 674).

Note that the size function is only used for marking cells *before* remeshing. The size function is not used to govern the size of the cell during remeshing.

For steady-state applications (see Steady-State Dynamic Mesh Applications (p. 744)), you can instruct ANSYS Fluent to perform a second round of cell marking and agglomeration after the boundary has moved, based on skewness criteria. The intent is to further improve the mesh quality through additional local remeshing. This optional feature works in conjunction with the Dynamic Mesh Task Page (p. 2471) (Figure 10.33: The Remeshing Tab in the Mesh Method Settings Dialog Box (p. 667)), and operates according to the skewness parameters you set in this dialog box. The size function parameters are not considered during this additional remeshing. Note that enabling this option will increase the time required to update the mesh during the solution.

Important

Additional local remeshing after the boundary has moved is not available for transient dynamic mesh applications, as the resulting numerical method would no longer be conservative.

When you use the **Sizing Function** remeshing option (see Figure 10.34: The Remeshing Tab in the Mesh Method Settings Dialog Box Using the Sizing Function Option (p. 668)), you can control three parameters that govern the size function. You can specify the **Size Function Resolution**, the **Size Function Variation**, and the **Size Function Rate** or you can return to ANSYS Fluent's default values by using the **Use Defaults** button.

The size function **Resolution** controls the density of the background mesh (see Local Remeshing Based on Size Functions (p. 670)). By default, it is equivalent to 3 in 2D simulations and 1 in 3D simulations.

The size function **Variation** corresponds to α in Equation 10.20 (p. 674). It is the measure of the maximum permissible cell size and it ranges from $-1 < \alpha < +\infty$.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The size function **Rate** corresponds to β in Equation 10.20 (p. 674). It is the measure of the rate of growth of the cell size, and it ranges from $-0.99 < \beta < 0.99$. A value of 0 implies linear growth, whereas higher values imply a slower growth near the boundary with faster growth as one moves toward the interior.

Note

To employ additional local remeshing, first make sure that you have enabled the **Remeshing** option in the Dynamic Mesh Task Page (p. 2471).

You can also use the following text command:

`define` → `dynamic-mesh` → `controls` → `remeshing-parameter` → `remeshing-after-moving`?

Finally, type `yes` to the question, `optional remeshing after moving the mesh`?

10.6.2.3.2. Cell Zone Remeshing Method

The cell zone remeshing method allows for the remeshing of the complete cell zone, and provides the option to also remesh the faces of all adjacent deforming dynamic face zones. This remeshing method is enabled by default when local cell remeshing is enabled, and is performed automatically if the local cell remeshing does not produce an acceptable mesh (see Local Remeshing Method (p. 668) for the acceptability criteria). Cell zone remeshing can also be manually invoked, using the following text command:

`define` → `dynamic-mesh` → `actions` → `remesh-cell-zone`

The cell zone remeshing method is available for triangular cells in 2D meshes and tetrahedral cells in 3D meshes. For 3D meshes, the method also allows the remeshing of wedge cells in boundary layers. The detection of the boundary layers (as well as the wedge element height distribution and number of layers) is automatically performed by default, and allows for different layer parameters on each boundary zone. You can manually specify the parameters by entering nonzero values for first height, growth rate, and number of layers via the text commands available in the `define/dynamic-mesh/controls/remeshing-parameters/prism-layer-parameters` menu, although generally it is not necessary to do this. When you enter them manually, the boundary layer parameters are global: they apply to every prism layer detected in the remeshing zone.

Note that it is necessary to enable the dynamic mesh model and the remeshing method in order to attain access to the prism layer controls, even if the cell zone is remeshed manually and not as part of a dynamic mesh update.

10.6.2.3.2.1. Limitations of the Cell Zone Remeshing Method

Zone remeshing has the following limitations:

- Only triangular, tetrahedral, and wedge cell types (when the wedge cells are part of a 3D boundary layer mesh) are remeshed.
- Zones with hanging nodes cannot be remeshed.

10.6.2.3.3. Face Region Remeshing Method

The face region remeshing method allows for the remeshing of those triangular faces (in 3D meshes) and linear faces (in 2D meshes) that are on a deforming face zone and adjacent to a moving face zone

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(see layer j in Figure 10.40: Expanding Cylinder Before Region Face Remeshing (p. 677)). ANSYS Fluent marks the faces based on minimum and maximum length scales, and then remeshes the faces and the associated cells to produce a very regular mesh on the deforming boundary. Although primarily designed for in-cylinder type configurations, where the remeshing region is located where cylinder walls meet the moving piston, face region remeshing can be used for all applications where a moving dynamic zone abuts deforming dynamic face zones.

For 3D simulations, ANSYS Fluent allows face region remeshing with symmetric boundary conditions and across multiple face zones. The remeshing can preserve features not only between the different deforming face zones, but also within a face zone. For more information on feature preservation, see Feature Detection (p. 690). ANSYS Fluent also allows face region remeshing of tetrahedral cell zones that contain wedge cells in boundary layers, as described in the section that follows.

To begin marking the faces for face region remeshing, ANSYS Fluent identifies the nodes at the intersection of a moving dynamic zone and the adjacent deforming zones. ANSYS Fluent then analyzes the height of the faces on the deforming zones that are in the range of the identified nodes, and then remeshes the faces depending on the specified maximum or minimum length scale.

Consider the simple tetrahedral mesh of a cylinder that has a moving end wall (see Figure 10.40: Expanding Cylinder Before Region Face Remeshing (p. 677)). The faces that are subject to remeshing are in layer j of the side wall. If the faces in layer j are expanding, the expansion continues until the height h reaches the maximum length scale, and then the layer is remeshed to form 2 layers of elements (see Figure 10.41: Expanding Cylinder After Region Face Remeshing (p. 678)). Conversely, if the faces of layer j are contracting, the contraction continues until h reaches the minimum length scale, and then layer j and the neighboring layer of faces (layer i) on the deforming zone are remeshed to form a single layer.

Figure 10.40: Expanding Cylinder Before Region Face Remeshing

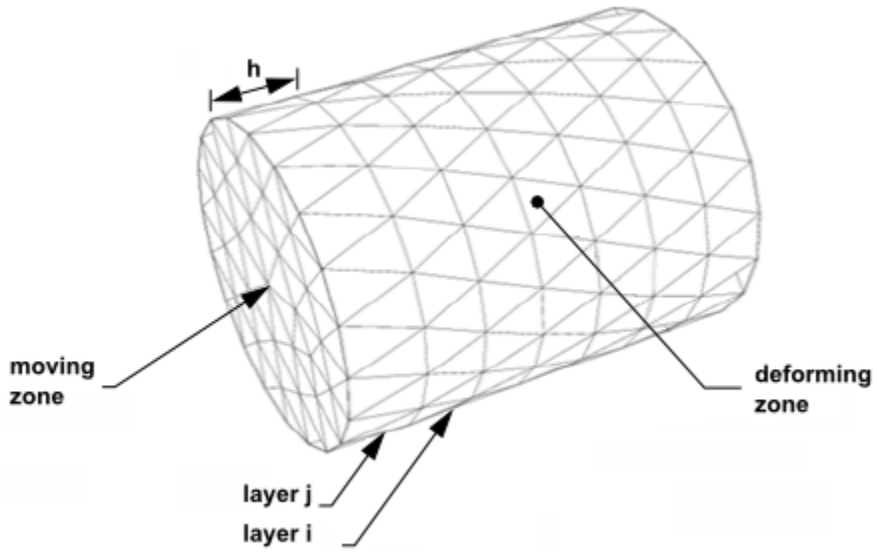
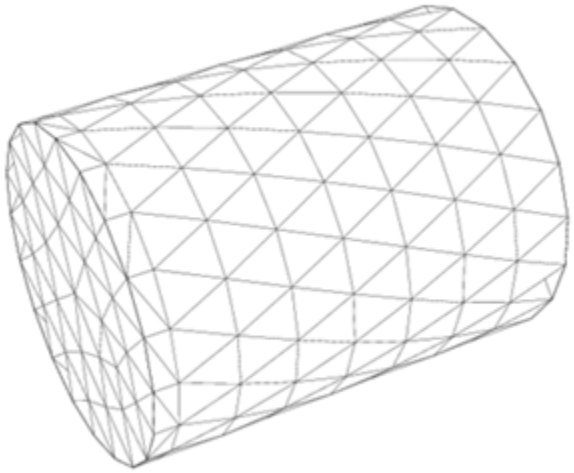


Figure 10.41: Expanding Cylinder After Region Face Remeshing



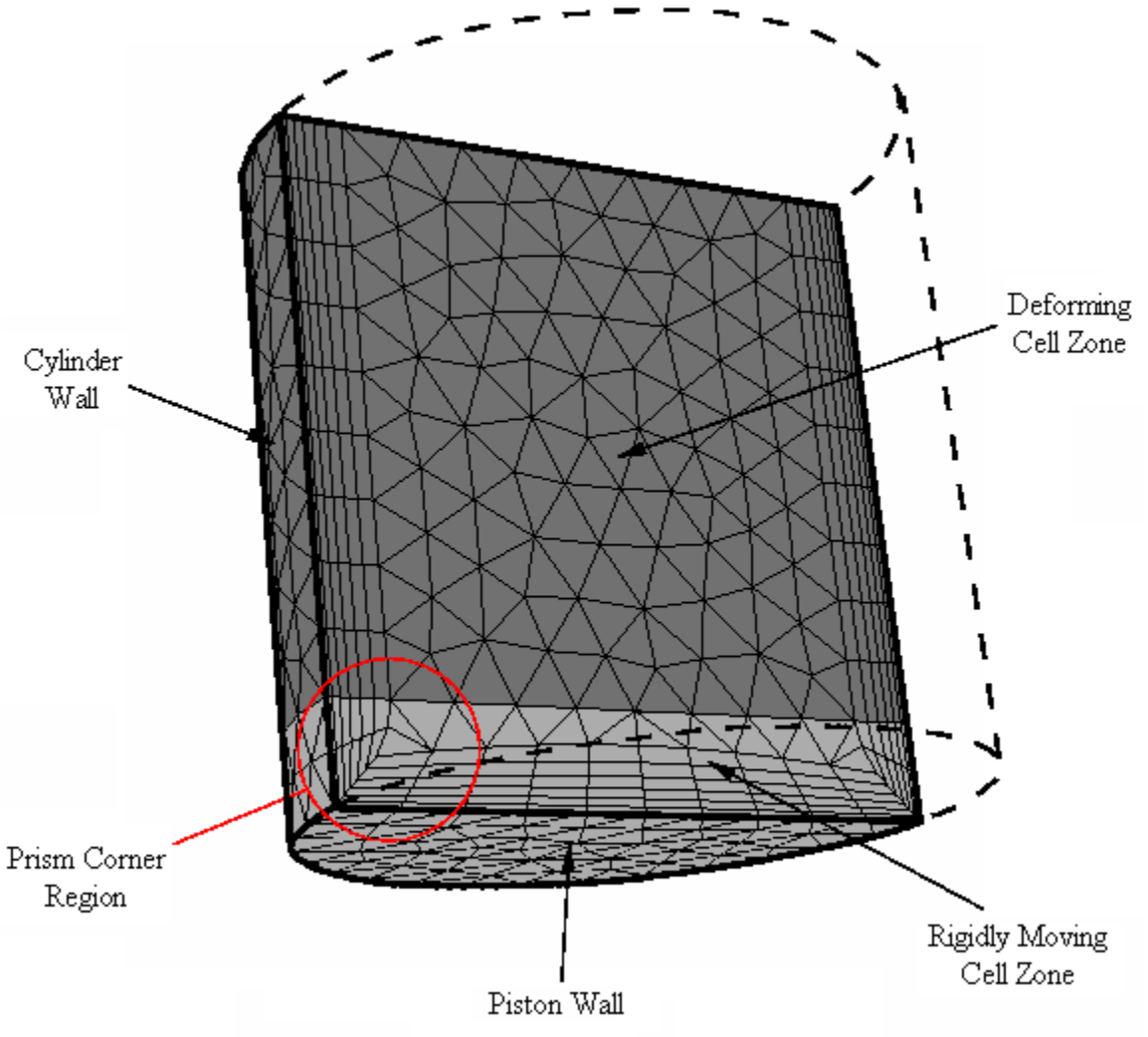
10.6.2.3.3.1. Face Region Remeshing with Wedge Cells in Prism Layers

In 3D simulations, the face region remeshing method can be applied on meshes that have wedge cells along the deforming face zones. When remeshing the faces on the deforming face zones, the associated wedge cells are remeshed as well. The layer parameters are based on the existing mesh by default; you have the option of manually setting these parameters, as described later in this section.

If the motion of the face zone is large compared to the height of the adjacent deforming face zones, it is recommended that you decompose the mesh volume in such a way as to create a dynamic cell zone that moves as a rigid body in between the moving face zone and the deforming dynamic cell zone on which face region remeshing is applied. Consider Figure 10.42: Volume Decomposition for Prism Layers (p. 679), which displays only half of an in-cylinder mesh. The rigidly moving cell zone encapsulates the prism layers on the moving piston so that the layers are not remeshed, and therefore the risk of generating degenerate cells during the mesh motion update is reduced.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.42: Volume Decomposition for Prism Layers

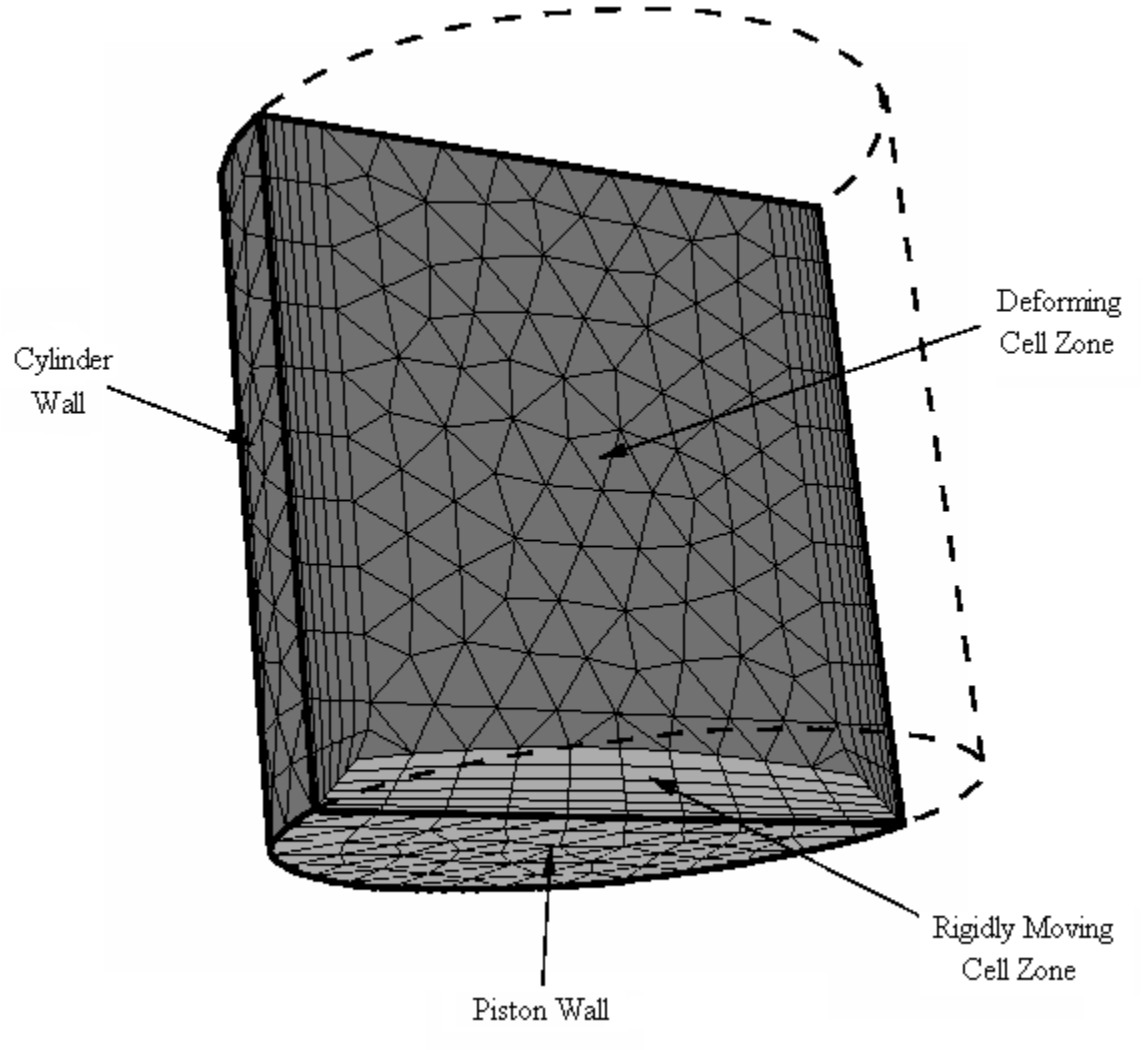


It is preferable (and even mandatory, if the mesh is a half model with a symmetry plane) to decompose the volume such that the "corner" region of the prism layers (shown in the previous figure) exists entirely within the rigidly moving zone. This allows for the largest deformations without risking degenerate elements, because the prism normals of the remeshed cells are uniformly perpendicular to the faces undergoing remeshing.

If the range of motion does not allow you to encapsulate the entire corner region of the prism layers in a rigidly moving zone, it is recommended that you encapsulate the "base" of the prism layers (shown in Figure 10.43: Volume Decomposition for the Base of the Prism Layers (p. 680)) and move these cells with a rigid body motion. Although this is less ideal than encapsulating the corner region, it does reduce the risk of degenerate mesh elements.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.43: Volume Decomposition for the Base of the Prism Layers



For piston-type applications that contain prism layers, a reasonable rule of thumb is that you should decompose the mesh volume if the piston motion is more than half the cylinder height. If you decide not to decompose the volume at all, you must at the very least enable the **Deform Adjacent Boundary Layer with Zone** option in the **Meshing Options** tab of the **Dynamic Mesh Zone** dialog box when setting up the moving face zone (see Rigid Body Motion (p. 726) for details). In any case, it is recommended that you always preview the mesh motion over the complete simulation time, to make sure that you will have a valid mesh at each time step.

The prism layer parameters (that is, element height, growth rate, and number of layers) are extracted automatically from the mesh and do not generally require your input. To prevent the prism parameters from drifting due to repeated remeshing, the prism parameters for first height, growth rate, and number of layers can be entered manually, using the text commands available in the `define/dynamic-mesh/controls/remeshing-parameters/prism-layer-parameters` menu.

10.6.2.3.3.2. Applicability of the Face Region Remeshing Method

Note the following limitations associated with face region remeshing:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You can use the face region remeshing method only in cell zones that contain triangular cells (in 2D) or tetrahedral cells, with or without prism layers (in 3D). For 3D meshes, the faces on the deforming boundaries that border the moving face must all be triangular.
- The face region remeshing method is not compatible with diffusion-based smoothing or linearly elastic solid smoothing.
- You cannot use the face region remeshing method in conjunction with hanging node adaption. For more information on hanging node adaption, see Hanging Node Adaption in the Theory Guide.

10.6.2.3.4. CutCell Zone Remeshing Method

The CutCell zone remeshing method is available to remesh a complete cell zone (including all boundary zones of the remeshed cell zone). This method is available for 3D simulations only. The existing volume mesh is replaced by a predominantly Cartesian mesh. Inflation layers can be added as an option. When used as part of a transient simulation, the remeshing occurs at predefined intervals and whenever the mesh quality in the cell zone is deemed poor. Examples of meshes before and after CutCell zone remeshing are shown in Figure 10.44: Unstructured Tetrahedral Mesh Before CutCell Zone Remeshing (p. 681) and Figure 10.45: Mesh After CutCell Zone Remeshing (p. 682), respectively. Figure 10.46: CutCell Zone Remeshing With Inflation Layers (p. 682) shows a cut through a CutCell mesh with inflation layers.

Figure 10.44: Unstructured Tetrahedral Mesh Before CutCell Zone Remeshing

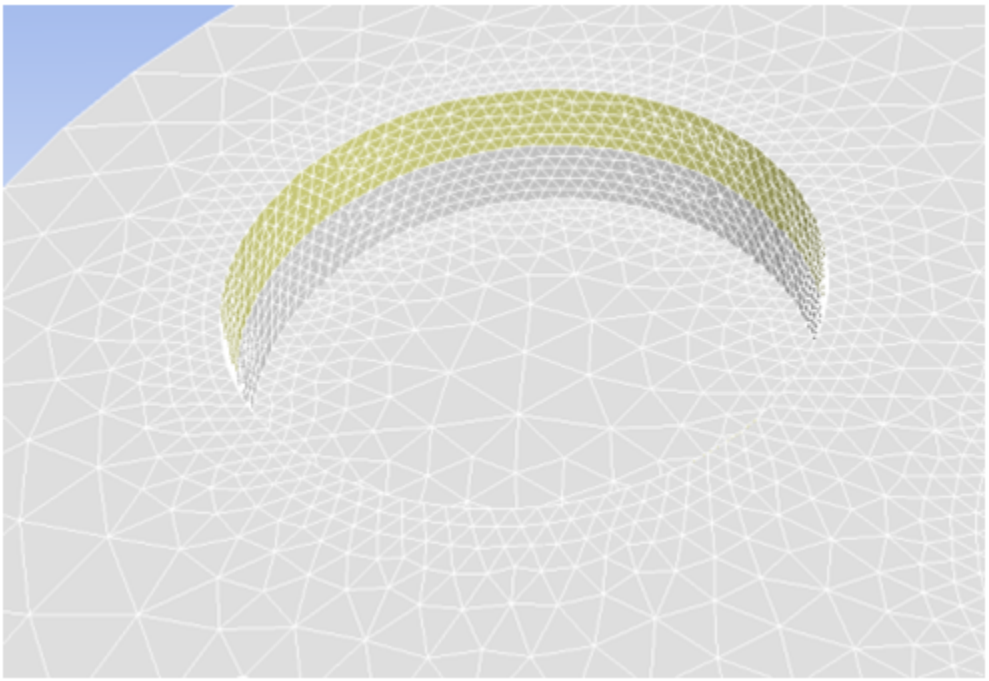


Figure 10.45: Mesh After CutCell Zone Remeshing

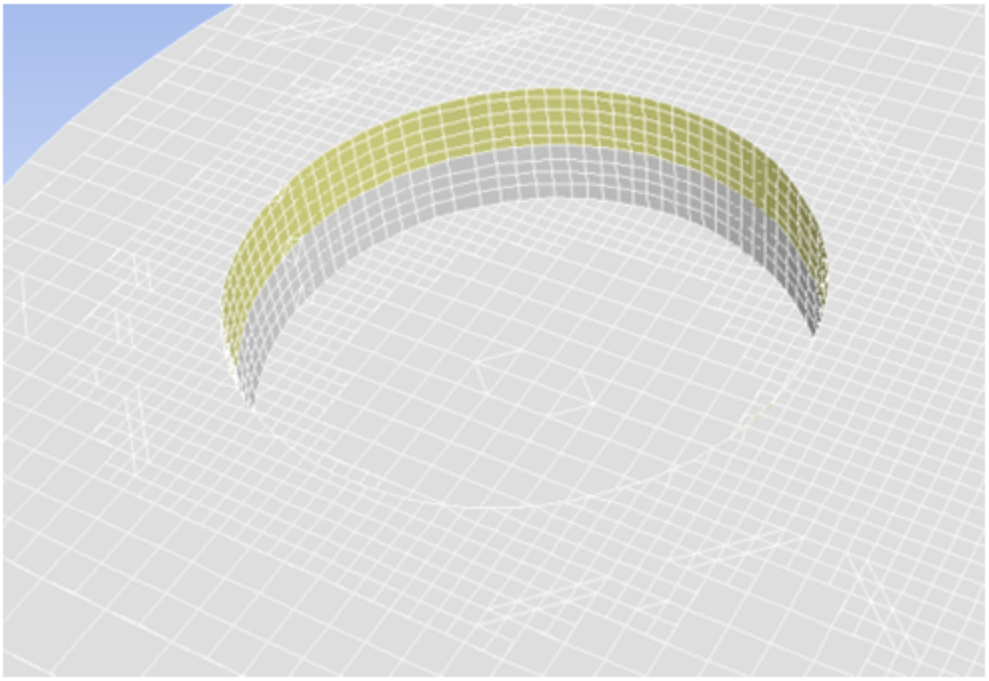
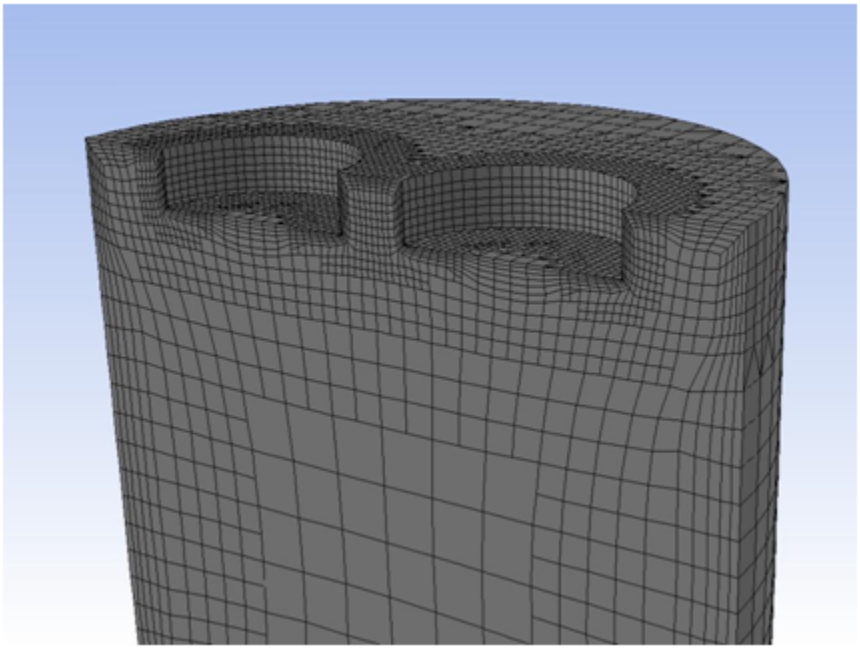


Figure 10.46: CutCell Zone Remeshing With Inflation Layers



During CutCell zone remeshing, a uniform Cartesian grid is locally refined using size functions, in order to resolve the initial mesh with sufficient accuracy. The resulting mesh consists mostly of hexahedral elements, which reduces the cell count compared to unstructured tetrahedral meshing. Additional element types are used near the boundaries to closely resolve complex shapes. The remeshing takes place at regular intervals, or whenever the mesh quality deteriorates due to mesh motion.

The CutCell zone remeshing method not only replaces the volume mesh, it also replaces the complete surface mesh of the remeshed cell zone. Consequently, this method can only be used to remesh cell zones that are either stand-alone zones or are only connected to other cell zones through non-conformal

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

interfaces. If inflation layers are added to a CutCell mesh, the inflation layers are grown from the remeshed surface mesh, and the CutCell mesh is morphed such that it smoothly matches the cap surfaces of the inflation layers.

The **CutCell Zone** remeshing method option is available in the **Remeshing Methods** group box. Note that the **CutCell Zone** remeshing method does not use any of the parameters specified in the **Mesh Method Settings** dialog box. The parameters used to control **CutCell Zone** remeshing are specified in the **Dynamic Mesh Zones** dialog box when setting up the zone and boundaries.

For more information about CutCell meshes, see the Fluent Meshing User's Guide.

10.6.2.3.4.1. Applicability of the CutCell Zone Remeshing Method

The CutCell zone remeshing method can be applied to cell zones, with the following limitations:

- The case must be 3D.
- The cell zone cannot be conformally connected to other cell zones; that is, a CutCell zone must be a stand-alone cell zone, or can only be connected to another zone through a non-conformal interface. In the latter case, the non-conformal interface is cleared before the remeshing and automatically recreated after the remeshing.

Note that the CutCell zone remeshing method can be applied to cell zones that contain adapted cells.

In parallel, all cells of the remeshed zone will be automatically migrated to and remeshed on a single CPU. Consequently, the machine memory will limit the size of the zone that can be remeshed. The mesh is automatically repartitioned after the remeshing.

In addition to the limitations listed previously, you should also note the following with regard to CutCell zone remeshing:

- Internal coupled walls, such as baffles, are discarded during the remeshing. The method is designed to primarily remesh cell zones with a single interior zone.
- Interior jump boundary condition zones (for example, fans, porous jumps), are discarded during the remeshing.
- Conformal periodic boundary conditions are not maintained after the remeshing. Any existing conformal periodic boundaries on the zone being remeshed should slit and replaced with non-conformal periodic boundaries.
- If the zone being remeshed contains multiple interior zones prior to the remeshing, all interior zones will be collected into a single interior zone during the CutCell zone remeshing.

10.6.2.3.4.2. Using the CutCell Zone Remeshing Method

In order to apply CutCell zone remeshing to a cell zone, begin by enabling the **CutCell Zone** option in the **Remeshing Methods** group box in the **Remeshing** tab of the **Mesh Method Settings** dialog box (see Figure 10.33: The Remeshing Tab in the Mesh Method Settings Dialog Box (p. 667)). Next, use the **Dynamic Mesh Zones** dialog box to create a deforming dynamic zone for the cell zone (see Deforming Motion (p. 731)). Then enable **CutCell** in the **Remeshing Options** group box and enter the relevant remeshing parameters, including: the maximum mesh size for the Cartesian cells; the global size function growth rate; the minimum orthogonal quality and remeshing interval to control the remeshing frequency; and the optional global inflation layer settings.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS Fluent allows you to specify either a soft or a mesh-based size function to control how the Cartesian mesh increases in size from the boundary toward the interior of the cell zone. For the soft size function, you specify a maximum mesh size for each boundary zone of the CutCell cell zone. When you use the mesh-based size function, ANSYS Fluent analyzes the existing mesh to evaluate the necessary mesh refinement at the boundary. See step 4 of Stationary Zones (p. 723) for more information about these size functions. You have control over the size function types and parameters by defining the boundary zones as dynamic zones. If no boundary zone adjacent to the CutCell zone is defined as a dynamic zone, ANSYS Fluent will automatically apply mesh-based size functions to each CutCell boundary zone and use the global growth rate entered for the cell zone to control the remeshing.

Note that the size functions used by the CutCell zone remeshing method are unrelated to the size function used for local cell remeshing (as described in Local Remeshing Based on Size Functions (p. 670)).

ANSYS Fluent allows you to add inflation layers to a CutCell mesh in order to better resolve wall-bounded flow features. After enabling **Inflation Layers** under the **CutCell Zone Parameters** for the CutCell cell zone, you can specify the global inflation parameters in the **Inflation Settings** dialog box. There are two types of inflation layers available: a **constant** type, where you can specify the constant height of the first layer; and an **aspect-ratio** type, where the first layer height is locally evaluated based on the specified aspect ratio. The latter is recommended and used by default since it is more robust and generally produces higher quality CutCell meshes. By default, the inflation settings specified apply to every boundary of the cell zone. If you would like to specify different inflation settings for different boundaries, you can define these boundaries as dynamic zones and enable **Zonal Inflation Layer Control**. Note that the **Number of Layers**, other than 0, has to be the same on all boundary zones. If different numbers of layers are specified on different boundaries, ANSYS Fluent will only inflate up to the largest common number specified. By setting the number of layers on a boundary zone to 0, you can locally suppress the growth of inflation layers, however, note that the transition is handled by stair stepping and adding wedge elements, which tends to generate skewed elements for small layer element heights.

The cells that contain hanging nodes / edges as a result of the CutCell zone remeshing are automatically converted to polyhedral cells. See Limitations (p. 144) for details about limitations associated with polyhedral cells.

It is recommended that you first manually apply the CutCell zone remeshing method (as described in the section that follows) and inspect the mesh before using the method as part of a transient simulation, as this allows you to evaluate if your remeshing parameters are suitable before running a calculation that is computationally expensive.

10.6.2.3.4.3. Applying the CutCell Zone Remeshing Method Manually

Although the CutCell zone remeshing method is primarily intended as an automatic remeshing method during dynamic mesh updates, you can also manually create a CutCell mesh (that is, you can remesh without running a calculation) by using the following text command:

`define` → `dynamic-mesh` → `actions` → `remesh-cell-zone-cutcell`

You will be prompted to enter the global parameters for the CutCell zone remeshing. If no dynamic zones are set up in your case, ANSYS Fluent will use the global parameters you enter and apply mesh-based size functions to each boundary of the CutCell zone. If the case is set up as a dynamic mesh case, manual remeshing will use the information specified on each CutCell boundary zone, including the zonal inflation layer settings. During manual CutCell zone remeshing, the global parameters are taken from the text interface input, and the global parameters specified on the CutCell cell zone are ignored.

When executing manual remeshing through the `remesh-cell-zone-cutcell` text command, you will be prompted to specify whether topology warnings should be ignored. If you do not ignore the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

topology warnings, ANSYS Fluent will analyze the zone being remeshed and report any interior face zones that will get discarded during the remeshing. If the zone being remeshed has periodic boundary zones, you will also be prompted to either abort the remeshing process or slit the periodics in order to proceed.

You will also be prompted to specify whether the cells that contain hanging nodes / edges as a result of the CutCell zone remeshing should be converted to polyhedral cells. See Limitations (p. 144) for details about limitations associated with polyhedral cells. By default, all hanging node cells are converted to polyhedral cells.

10.6.2.3.5. 2.5D Surface Remeshing Method

The 2.5D surface remeshing method only applies to extruded 3D geometries and is similar to local remeshing in two dimensions on a triangular surface mesh (not a mixed zone). Faces on a deforming boundary are marked for remeshing based on face skewness, minimum and maximum length scale; the 2.5D remeshing method also gives you the option of marking cells using size functions, as described in Local Remeshing Based on Size Functions (p. 670).

Figure 10.47: Close-Up of 2.5D Extruded Flow Meter Pump Geometry Before Remeshing and Laplacian Smoothing

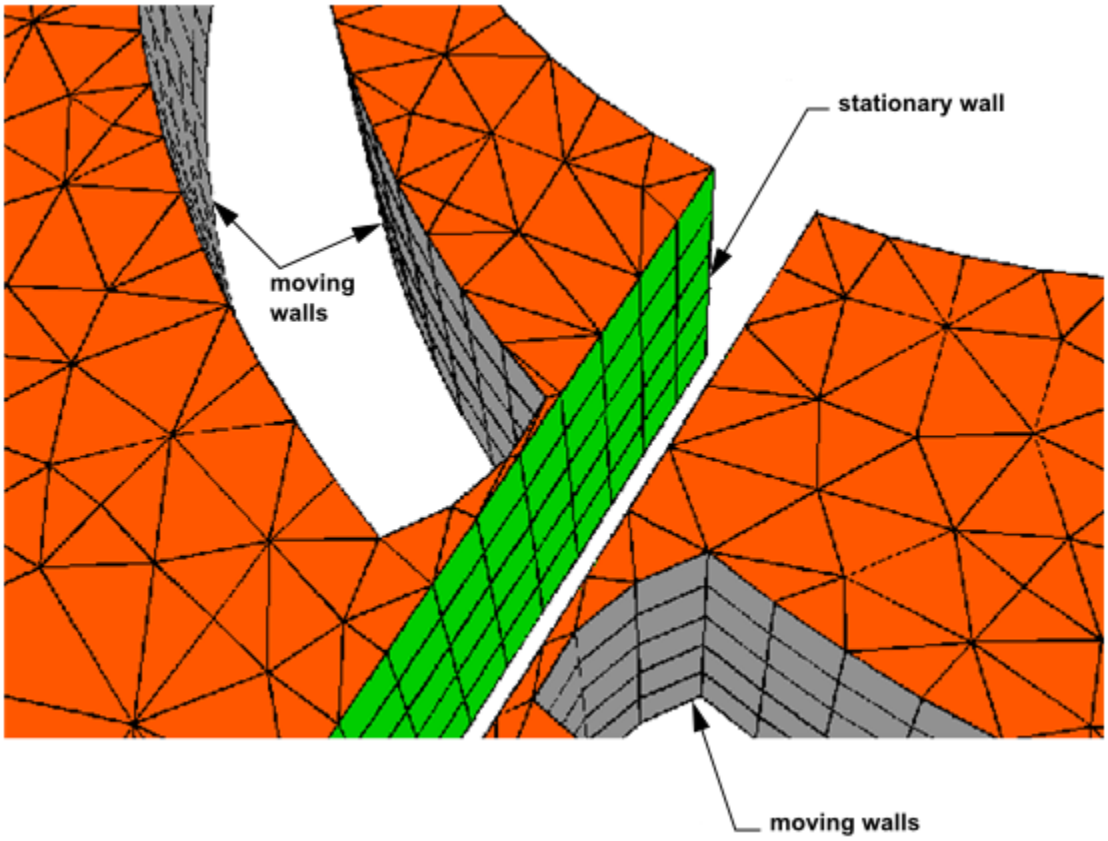
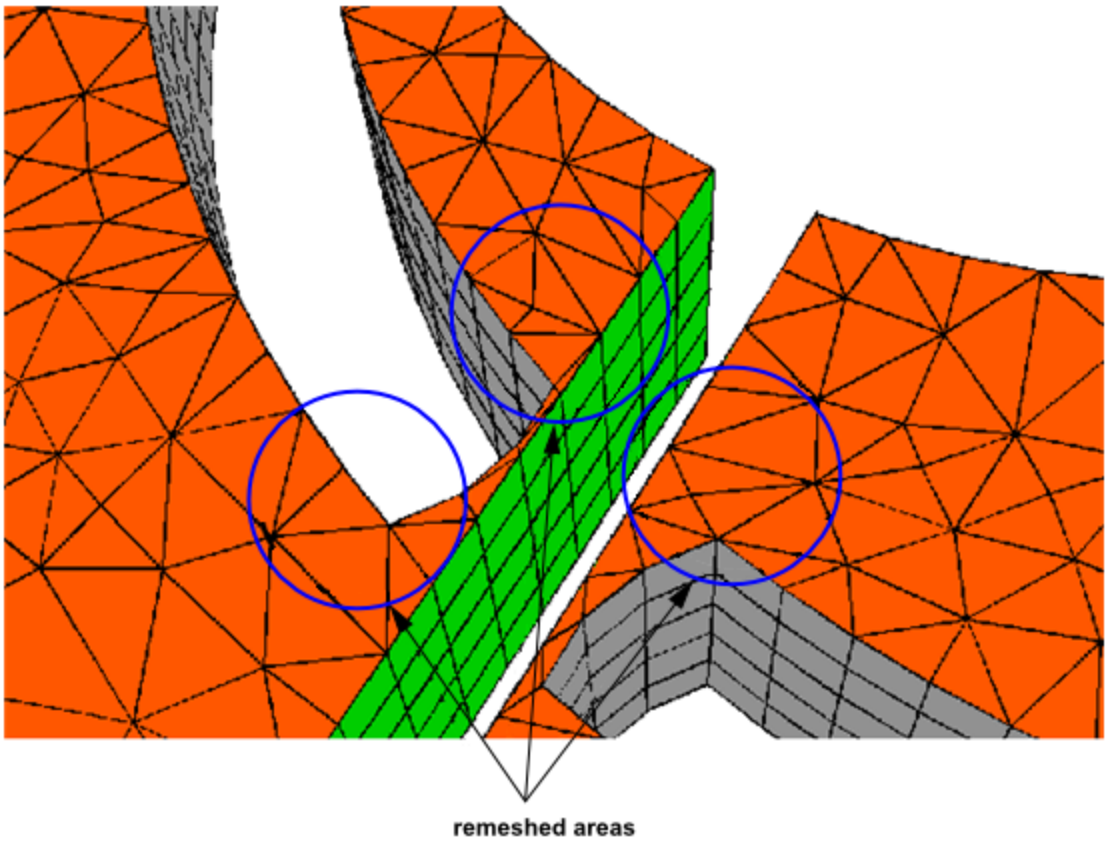


Figure 10.48: Close-Up of 2.5D Extruded Flow Meter Pump Geometry After Remeshing and Laplacian Smoothing



10.6.2.3.5.1. Applicability of the 2.5D Surface Remeshing Method

The following applies to the 2.5D surface remeshing method:

- Triangular faces get remeshed based on marking.
- Extruded wedges get remeshed based on the remeshing of the triangular face. Only extruded regions get remeshed, not mixed regions.
- The 2.5D remeshing method does not support remeshing or moving nodes on the perimeter of the extruded zone(s).
- Note that you cannot use the 2.5D surface remeshing method in conjunction with hanging node adaption. For more information on hanging node adaption, see Hanging Node Adaption in the Theory Guide.
- The extrusion must be along a straight line normal to the deforming zone and the cross-section of the extruded/coopered mesh must be constant along the extrusion direction. For more information about the 2.5D model, see Using the 2.5D Model (p. 686).
- Periodics are not supported at the extruded zones.

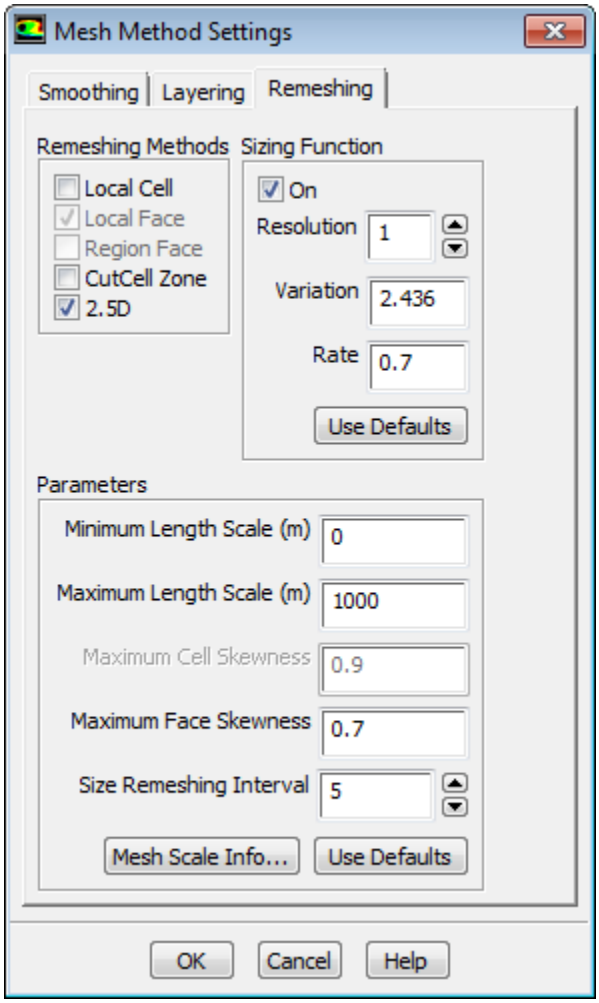
10.6.2.3.5.2. Using the 2.5D Model

For 3D simulations only, you can select the **2.5D** model under the **Remeshing** tab in the Mesh Method Settings Dialog Box (p. 2474). This model allows for a specific subset of remeshing techniques.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The 2.5D mesh essentially is a 2D triangular mesh which is expanded, or extruded, along the normal axis of the specific dynamic zone that you are interested in modeling. The triangular surface mesh is remeshed and smoothed on one side, and the changes are then extruded to the opposite side. Rigid body motion is applied to the moving face zones, while the triangular extrusion surface is assigned to a deforming zone with remeshing and smoothing enabled. The opposite side of the triangular mesh is assigned to be a deforming zone as well, with only smoothing enabled, as in Figure 10.50: 2.5D Extruded Gear Pump Geometry (p. 688).

Figure 10.49: The Remeshing Tab for the 2.5D Model

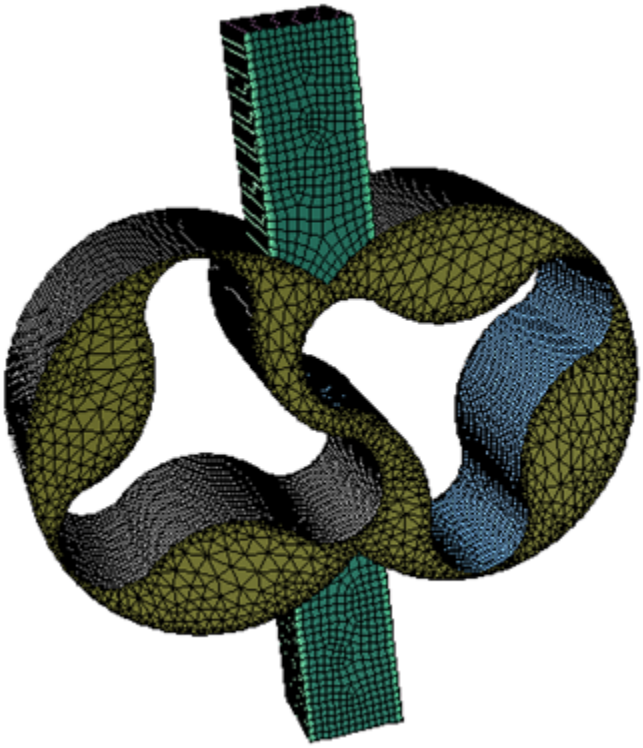


For more information on setting smoothing and remeshing parameters, see Dynamic Mesh Update Methods (p. 644).

The 2.5D model only applies to mappable (that is, extrudable) mesh geometries such as pumps, as in Figure 10.50: 2.5D Extruded Gear Pump Geometry (p. 688). Only the aspects of the geometry that represent the “moving parts” must be extruded in the mesh.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.50: 2.5D Extruded Gear Pump Geometry



Important

You must only apply smoothing to the opposite side of the extruded mesh, since ANSYS Fluent requires the geometry information for the dynamic zone. ANSYS Fluent projects the nodes back to its geometry after the extrusion. Without this geometry information, the dynamic zones tends to lose its integrity.

Important

In parallel, a partition method that partitions perpendicular to the extrusion surface should be used. For example, if the normal of the extrusion surface points in the X-direction then Cartesian-Y or Cartesian-Z would be the perfect partition methods.

The 2.5D model is used in combination with a `DEFINE_GRID_MOTION` UDF. (See Hooking `DEFINE_GRID_MOTION` UDFs in the Fluent Customization Manual for information about hooking this UDF.)

This UDF is associated with the extrusion surface that is adjacent to the cell zone, in turn applying the same deformation to the entire cell zone. This approach is particularly useful when modeling gear pumps that are predominantly extruded hexahedral meshes. For more information about this UDF, contact your support engineer.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.6.2.4. Volume Mesh Update Procedure

The volume mesh is updated automatically based on the methods described in Dynamic Mesh Update Methods (p. 644). ANSYS Fluent decides which method to use for a particular zone based on which model is enabled and the shape of the cells in the zone. For example, if the boundaries of a tetrahedral cell zone are moving, and unless the zone has been set up for CutCell zone remeshing, the mesh smoothing and local remeshing methods will be used to update the volume mesh in this zone. If the zone consists of prismatic (hexahedral and/or wedge) cells, then the dynamic layer method will be used to determine where and when to insert and remove cell layers. On extruded prism zones, the 2.5D surface meshing method will be used.

Depending on which model is enabled, ANSYS Fluent automatically determines which method to use by visiting the adjacent cell zones and setting appropriate flags for the volume mesh update methods to be used. If you specify the motion for a cell zone, ANSYS Fluent will visit all of the neighboring cell zones and set the flags appropriately. If you specify the motion of a boundary zone, ANSYS Fluent will analyze only the adjacent cell zones. If a cell zone does not have any moving boundaries, then no volume mesh update method will be applied to the zone.

Important

Note that as a result of the remeshing procedures, updated meshes may be slightly different when dynamic meshes are used in parallel ANSYS Fluent, and therefore very small differences may arise in the solutions.

Important

Note that if your dynamic mesh model consists of numerous shell conduction zones, the mesh update may be very time consuming because all shells are deleted and recreated during the mesh update.

10.6.2.5. Transient Considerations for Remeshing and Layering

The second order in time transient formulation can be used with remeshing, layering, and smoothing cases. In such cases, second order in time accuracy is preserved during layering events and/or mesh smoothing. However, during remeshing events, the time advancement accuracy reverts to first order. Therefore, it is recommended that you only use second order in time for cases with very low remeshing frequency, that is, for cases where remeshing occurs every several timesteps or more. If remeshing occurs every timestep, the method reduces to a first order formulation. For cases that need very frequent remeshing, it is recommended that you retain the first order in time formulation. For details, see Dynamic Mesh Theory in the *Fluent Theory Guide*.

The switch to first order happens automatically when the solver detects a topology change as a result of remeshing within a timestep. You will not see a change in the user interface, however you can monitor the switch between first and second order in time by using the following text command to enable the verbosity with dynamic meshes:

`define` → `dynamic-mesh` → `transient-settings` → `verbosity`

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A message showing the active time formulation will print to the console when verbosity is set to 1.

Note

For cases with second order in time with dynamic mesh enabled, any mesh swapping and manual mesh manipulation will trigger a switch to first order in time.

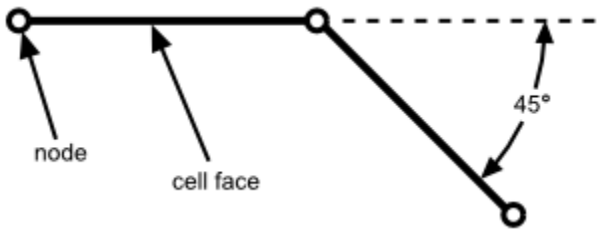
10.6.3. Feature Detection

The remeshing and/or smoothing of boundary zones may result in the chamfering of corners where cell faces meet. However, there may be some corners (identified by a particular angular range) that are important for your flow, and that you do not want chamfered. For 3D cases, you can specify that ANSYS Fluent detects such corners (or "features") and preserves them. Such features may be at the juncture of different boundary zones, or they may be on a single non-planar boundary zone.

In the **Geometry Definition** tab of the Dynamic Mesh Zones Dialog Box (p. 2490) for any **Definition** type, you can indicate whether you want to preserve features by enabling **Include Features** under **Feature Detection** and specifying a threshold value for the **Feature Angle**. A value of 180 degrees (the default) for the **Feature Angle** means that none of the corners must be preserved, whereas a value of 0 means that all of the corners must be preserved.

To illustrate the setting of the **Feature Angle**, see Figure 10.51: Cross Section of a 3D Corner (p. 690). If you set the **Feature Angle** to 40 degrees, this corner would be guaranteed to be preserved. If you set it to 50 degrees, this corner could be chamfered.

Figure 10.51: Cross Section of a 3D Corner



10.6.3.1. Applicability of Feature Detection

The following items are applicable for use with feature detection:

- Feature remeshing is only possible with face region remeshing.
- Features are preserved by local face remeshing, that is, there is no local face remeshing across features.
- Smoothing methods preserve features, that is, nodes at feature edges are not allowed to be smoothed.

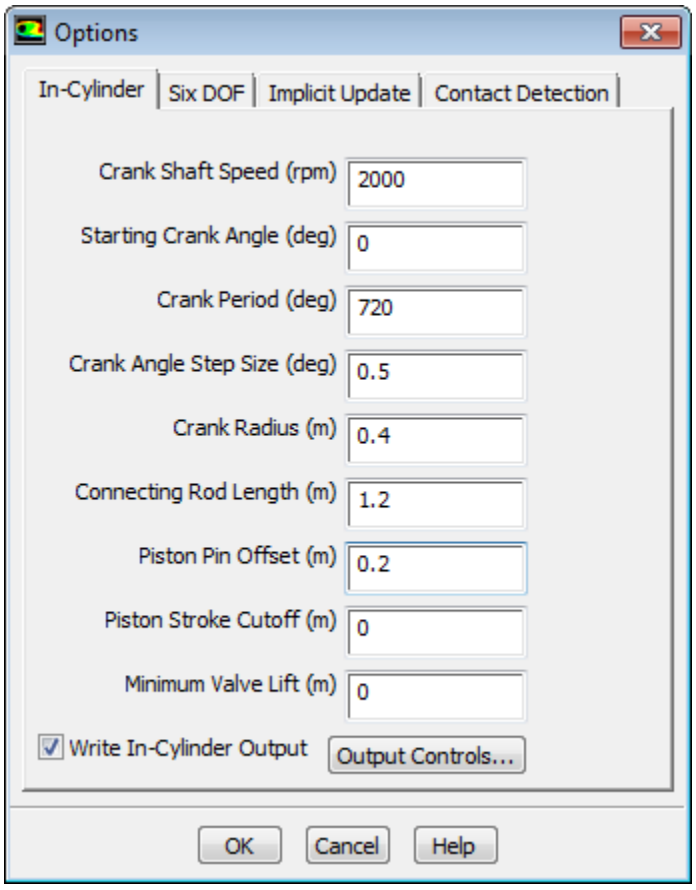
10.6.4. In-Cylinder Settings

You can enable the **In-Cylinder** option in the Dynamic Mesh Task Page (p. 2471) (Figure 10.13: The Dynamic Mesh Task Page (p. 643)) for transient problems. Then click the **Settings...** button in the **Options** group box to open the Options Dialog Box (p. 2478). Click the **In-Cylinder** tab and specify the **Crank Shaft Speed**, the **Starting Crank Angle**, and the **Crank Period**, which are used to convert between flow time and crank angle. You must also specify the time step to use for advancing the solution in

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

terms of crank angle in **Crank Angle Step Size**. By default, ANSYS Fluent assumes a **Crank Angle Step Size** of 0.5 degrees.

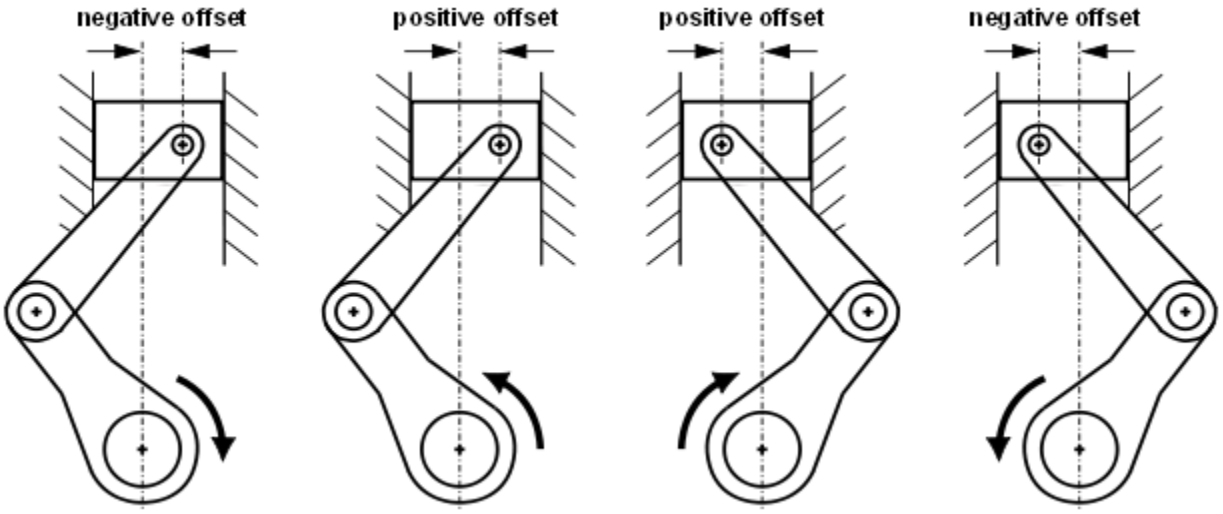
Figure 10.52: The In-Cylinder Tab of the Options Dialog Box



ANSYS Fluent provides a built-in function that can define the location of the piston as a function of crank angle. This function is named ****piston-full****, and is selected from the **Motion/UDF Profile** drop-down list in the **Dynamic Mesh Zones** dialog box as part of rigid body motion (see Rigid Body Motion (p. 726) for details). If you plan to specify the piston motion using this function, you must specify the **Crank Radius** (that is, the distance between the crank center and the center of the crank pin) and the **Connecting Rod Length**. You also have the option of entering a value for the **Piston Pin Offset** for cases when the piston pin is offset perpendicularly from the plane defined by the crank shaft axis and the direction of motion of the piston. The sign of this offset can be positive or negative, and is determined based on the geometry and the direction of rotation of the crank shaft (as shown in Figure 10.53: Determining the Sign of the Piston Pin Offset (p. 692)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.53: Determining the Sign of the Piston Pin Offset



When the ****piston-full**** function is used, the piston location is calculated according to the following equation:

$$p_s=\sqrt{(r_c+L)^2-x_{\text{offset}}^2}-r_c\cos(\theta_c)-\sqrt{L^2-(r_c\sin(\theta_c)+x_{\text{offset}})^2} \tag{10.25}$$

where p_s is the piston location, r_c the crank radius, L is the connecting rod length, x_{offset} is the piston pin offset, and θ_c is the current crank angle.

The piston location p_s is always 0 at top-dead-center (TDC), that is, when the crank pin is perfectly aligned between the piston pin and the center of rotation of the crank shaft. TDC occurs when the crank angle is 0° when there is no piston pin offset, and prior to the crank angle reaching 0° when there is a positive piston pin offset. The piston location is a positive value at bottom-dead-center (BDC), that is, when the crank shaft is perfectly aligned between the piston pin and the crank pin. The value of p_s at BDC is equal to $2r_c$ when there is no piston pin offset, and greater than $2r_c$ when there is a nonzero piston pin offset (positive or negative).

The current crank angle θ_c is calculated from

$$\theta_c=\theta_s+t\Omega_{\text{shaft}} \tag{10.26}$$

where θ_s is the **Starting Crank Angle** and Ω_{shaft} is the **Crank Shaft Speed**.

The **Piston Stroke Cutoff** and **Minimum Valve Lift** values are used to control the actual values of the valve lift and piston stroke such that

$$\begin{aligned} v_{lift}&=\max\left(v_{lift}^c, v_{lift}^{min}\right)\\ p_s&=\min\left(p_s^c, p_s^{min}\right)\end{aligned} \tag{10.27}$$

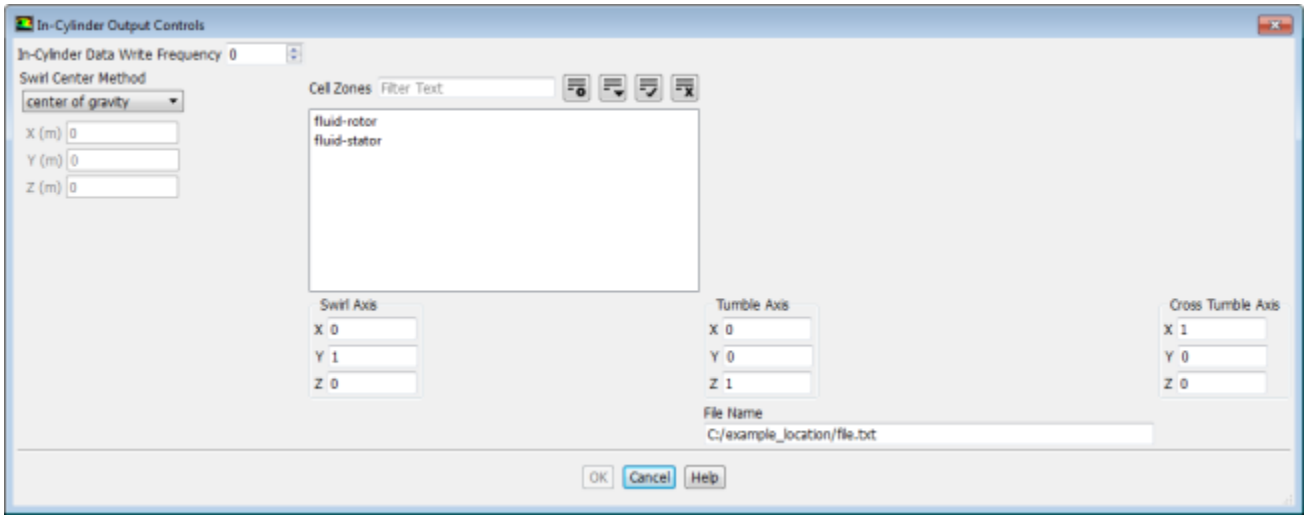
where v_{lift}^c is the valve lift computed from the appropriate valve profiles, v_{lift}^{min} is the **Minimum Valve Lift**, p_s^c is the stroke calculated from Equation 10.25 (p. 692), and p_s^{min} is the **Piston Stroke Cutoff**. (See Defining Motion/Geometry Attributes of Mesh Zones (p. 699) on how the **Piston Stroke Cutoff** is used to control the onset of layering in the cylinder chamber.)

Enable the **Write In-Cylinder Output** option then click the **Output Controls...** button if you want to specify specific output parameters. The In-Cylinder Output Controls Dialog Box (p. 2481) will open, where

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

you can specify various quantities needed for the calculation of swirl and tumble along with the frequency of writing the output to the chosen file. Swirl is used to describe circulation about the cylinder axis. Tumble flow circulates around an axis perpendicular to the cylinder axis, orthogonal to swirl flow.

Figure 10.54: The In-Cylinder Output Controls Dialog Box



The following list describes the In-Cylinder Output Controls Dialog Box (p. 2481).

In-Cylinder Data Write Frequency
is an integer entry specifying the interval in number of time-steps. Make sure that a value other than 0 is used for the frequency, in order to allow you to complete your setup.

Swirl Center Method
is a drop-down list that allows you to select the method to calculate the swirl center. The list contains **center of gravity** and **fixed**, with **center of gravity** being the default value.

center of gravity
option calculates the swirl center inside the code and is used as the center of gravity of the chosen cell zones.

fixed
option enables you to specify a swirl center in the entries below the drop-down list.

In addition to these two options, you can chose to use your own compiled UDF to calculate the swirl center.

For details on using a dynamic mesh UDF, see the Fluent Customization Manual for information on user-defined functions.

Cell Zones
is a list that displays the names of all existing cell zones in the case files. You can select only the zones relevant for the swirl and tumble calculations.

Swirl Axis
specifies the swirl axis with three entries for the directional components. By default, **X**, **Y**, **Z** = **0**, **1**, **0**.

Tumble Axis
specifies the directional components of **Tumble Axis** in **X**, **Y**, **Z** directions. By default, **X**, **Y**, **Z** = **0**, **0**, **1**. This applies only in 3D.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cross Tumble Axis

specifies the directional components of **Cross Tumble Axis** in **X**, **Y**, **Z** directions. By default, **X**, **Y**, **Z** = **1**, **0**, **0**. This applies only in 3D.

File Name

specifies the name of the **In-Cylinder** output file. By default, the file name contains the name of the case file appended with a `.txt` extension.

The **In-Cylinder** specific output controls can also be controlled using the TUI as follows:

```
  Go to
  define/dynamic-mesh/controls/in-cylinder-output?
 Enable in-cylinder output?[no] yes
 Output Write Frequency[0] 10
 Cell zone name/id(1)[()] 2
 Cell zone name/id(1)[()]
 File Name[''/nfs/devvault/data9/ic-sp-output.txt'']
 Swirl Center Method: (fixed cg user-defined)
  Option[cg]
  Swirl Axis x[0]
  Swirl Axis y[1]
  Swirl Axis z[0]
  Tumble Axis x[0]
  Tumble Axis y[0]
  Tumble Axis z[1]
  Cross Tumble Axis x[1]
  Cross Tumble Axis y[0]
  Cross Tumble Axis z[0]
```

If you select **fixed** as the choice at **Swirl Center Method** then you will be prompted to enter the swirl center as follows:

```
  Swirl Center(x) (mm) [0]
  Swirl Center(y) (mm) [0]
  Swirl Center(z) (mm) [0]
```

If a swirl center method UDF has been compiled already and loaded into UDF then you can choose **user-defined** as the swirl center method option, in such a case the following is the sequence of prompts.

```
  Swirl Center UDF[] swirl_udf::libudf
```

If the name of the UDF library is **libudf** then you can omit this and enter in the swirl center `UDF[]swirl_udf`, otherwise the name of the UDF followed by the UDF library name with symbol:: in between, should be entered.

By filling up the various entries that are needed in the In-Cylinder Output Controls Dialog Box (p. 2481) and pressing the **OK** button, the swirl and tumble calculations will be written at the chosen frequency to the chosen file while doing the solution run. Details of the quantities written to the file are as follows:

CA = Crank Angle

m = Mass of the entire fluid contained in the selected cell zones

L = Angular momentum vector of fluid mass contained in selected cell zones with respect to the swirl center

$|\vec{L}|$ = Magnitude of angular momentum of fluid

$\vec{sa}$ = Swirl Axis

$\vec{ta}$ = Tumble Axis

$\vec{cta}$ = Cross Tumble Axis

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$\mathbf{I}_{sa}$ = Moment of inertia of the fluid mass about Swirl axis

$\mathbf{I}_{ta}$ = Moment of inertia of the fluid mass about Tumble Axis

$\mathbf{I}_{cta}$ = Moment of inertia of the fluid mass about Cross Tumble Axis

$\cdot$ = Dot product between two vectors

Altogether, the previous quantities are combined to yield eight columns of data in the output file, as shown in the figure that follows:

Figure 10.55: Sample Output File Showing Various Quantities

CA	(L . sa)	(L . ta)	(L .cta)	|L|	I_{sa}	I_{ta}	I_{cta}
350.00	0.0000e+00	0.0000e+00	0.0000e+00	0.0000e+00	1.1474e-07	7.0881e-08	6.8234e-08
365.00	6.4910e-07	7.6332e-07	-2.3923e-08	1.0023e-06	9.9355e-08	6.1951e-08	5.9120e-08
380.00	-1.6684e-06	-2.3585e-06	-7.8704e-08	2.8900e-06	1.5460e-07	9.2601e-08	9.2951e-08
395.00	-3.0125e-05	8.1780e-06	2.9730e-06	3.1357e-05	2.9557e-07	1.7709e-07	1.8031e-07
410.00	-8.9259e-05	1.7637e-05	-1.3191e-05	9.1936e-05	4.9396e-07	3.1077e-07	3.1660e-07
425.00	-2.1336e-04	2.4657e-05	-4.6908e-05	2.1984e-04	7.3845e-07	5.1056e-07	5.1811e-07
440.00	-3.9555e-04	6.6114e-05	-1.0156e-04	4.1370e-04	1.0084e-06	7.8920e-07	7.9931e-07
455.00	-6.0621e-04	1.2651e-04	-1.7990e-04	6.4487e-04	1.2833e-06	1.1499e-06	1.1623e-06
470.00	-8.1472e-04	2.0109e-04	-2.6251e-04	8.7927e-04	1.5486e-06	1.5784e-06	1.5913e-06
485.00	-9.9456e-04	2.7342e-04	-3.6243e-04	1.0933e-03	1.7921e-06	2.0409e-06	2.0526e-06
500.00	-1.1160e-03	2.9711e-04	-4.5477e-04	1.2412e-03	2.0003e-06	2.4850e-06	2.4945e-06

10.6.4.1. Using the In-Cylinder Option

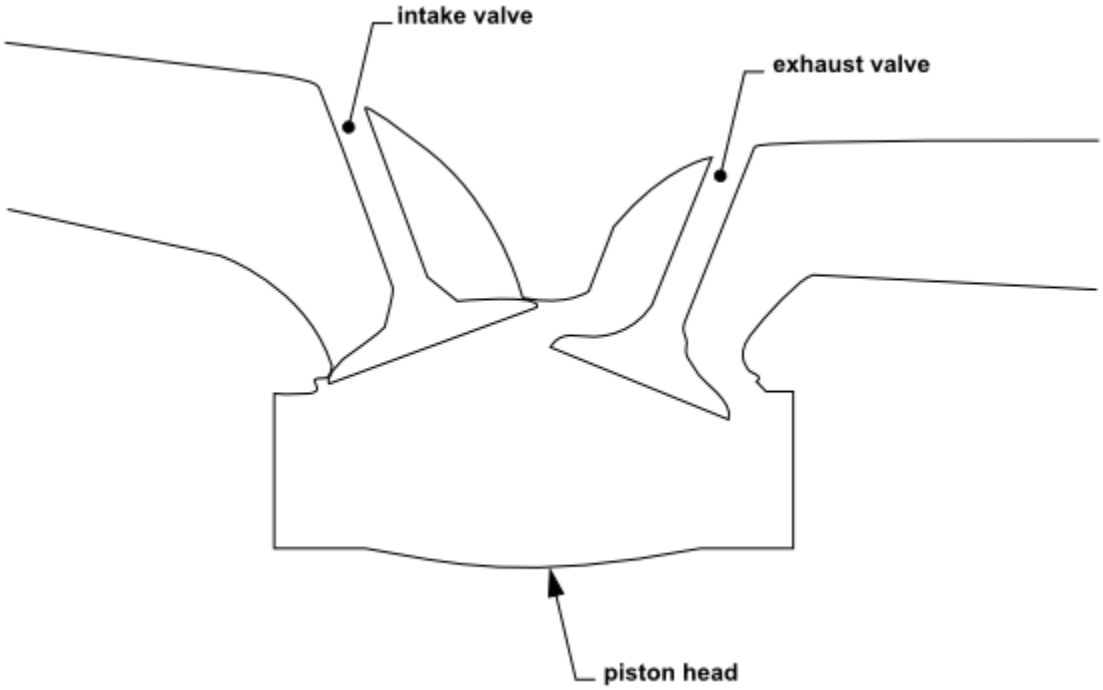
This section describes the problem setup procedure for an in-cylinder dynamic mesh simulation.

10.6.4.1.1. Overview

Consider the 2D in-cylinder example shown in Figure 10.56: A 2D In-Cylinder Geometry (p. 696) for a typical pent-roof engine.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.56: A 2D In-Cylinder Geometry



In setting up the dynamic mesh model for an in-cylinder problem, you must consider the following issues:

- how to provide the proper mesh topology for the volume mesh update methods (smoothing, dynamic layering, and local or zonal remeshing)
- how to define the motion attributes and geometry for the valve and piston surfaces
- how to address the opening and closing of the intake and exhaust valves
- how to specify the sequence of events that controls the in-cylinder simulation

10.6.4.1.2. Defining the Mesh Topology

ANSYS Fluent requires that you provide an initial volume mesh with the appropriate mesh topology such that the various mesh update methods described in Dynamic Mesh Update Methods (p. 644) can be used to automatically update the dynamic mesh. However, ANSYS Fluent does not require you to set up all in-cylinder problems using the same mesh topology. When you generate the mesh for your in-cylinder problem, you must consider the various mesh regions that you can identify as moving, deforming, or stationary, and generate these mesh regions with the appropriate cell shape.

The mesh topology for the example problem in Figure 10.56: A 2D In-Cylinder Geometry (p. 696) is shown in Figure 10.57: Mesh Topology Showing the Various Mesh Regions (p. 697), and the corresponding volume mesh is shown in Figure 10.58: Mesh Associated With the Chosen Topology (p. 697).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.57: Mesh Topology Showing the Various Mesh Regions

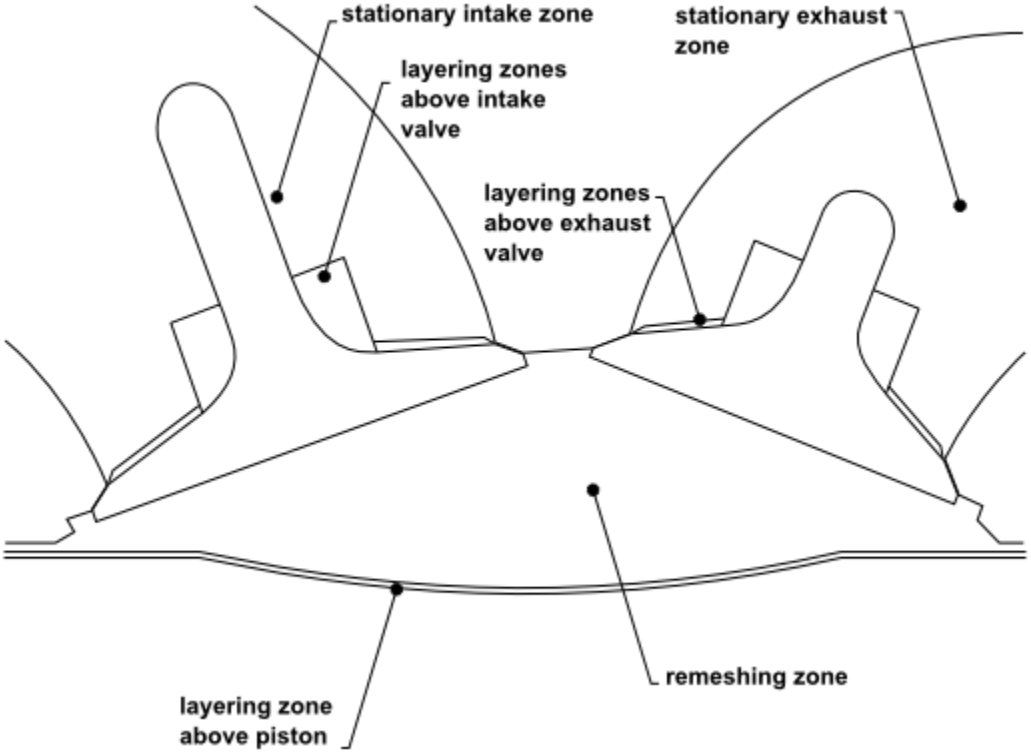
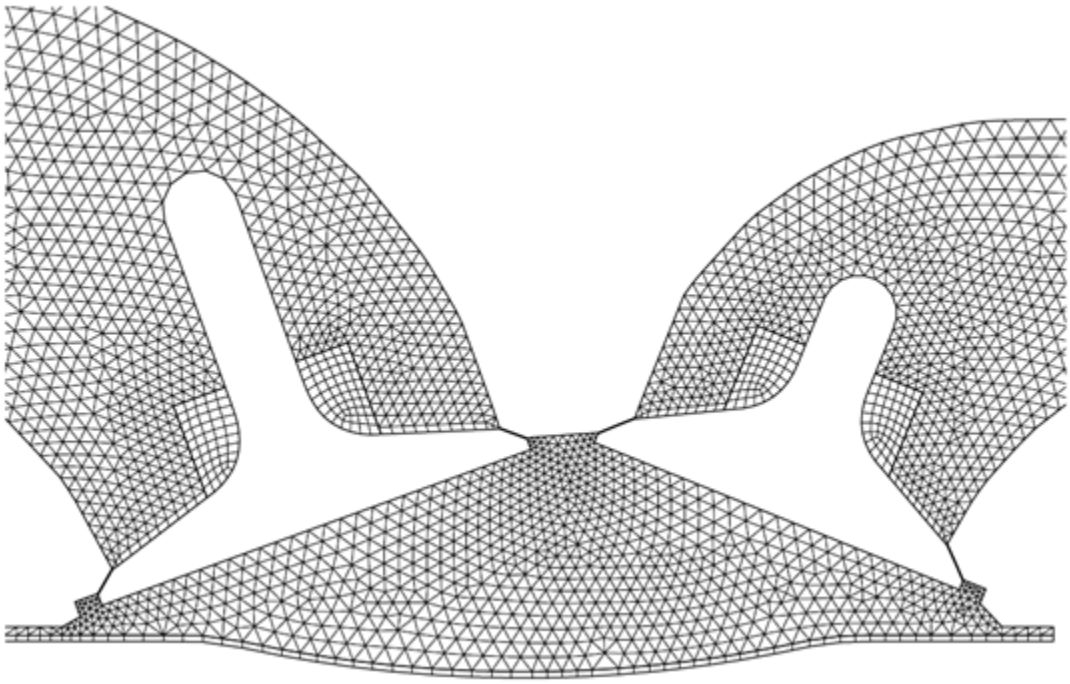


Figure 10.58: Mesh Associated With the Chosen Topology



Because of the rectilinear motion of the moving surfaces, you can use dynamic layering zones to represent the mesh regions swept out by the moving surfaces. These regions are the regions above the top surfaces of the intake and exhaust valves and above the piston head surface, and must be meshed with quadrilateral or hexahedral cells (as required by the dynamic layering method).

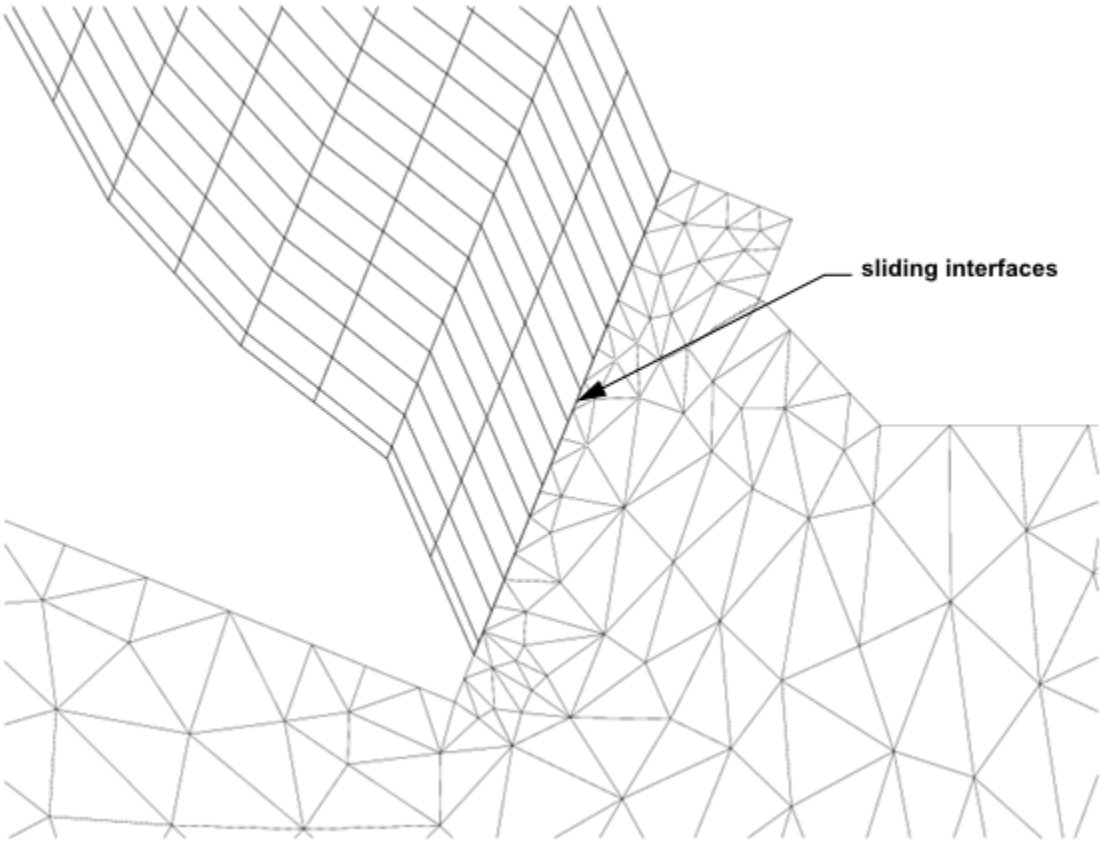
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For the chamber region, you must define a remeshing zone (triangular cells) to accommodate the various positions of the valves in the course of the simulation. In this region, the motion of the boundaries (valves and piston surfaces) is propagated to the interior nodes through smoothing. If the cell quality violates any of the remeshing criteria that you have specified, ANSYS Fluent will automatically agglomerate these cells and remesh them. Furthermore, ANSYS Fluent will also remesh the deforming faces (based on the minimum and maximum length scale that you have specified) on the cylinder walls as well as those on the sliding interfaces used to connect the chamber cell zone to the layering zones above the valve surfaces.

For the intake and exhaust port regions, you can use either triangular or quadrilateral cell zones because these zones are not moving or deforming. ANSYS Fluent will automatically mark these regions as stationary zones and will not apply any mesh motion method on these cell zones.

The dynamic layering regions above the piston and valves are conformal with the adjacent cell zone in the chamber and ports, respectively, so you do not have to use sliding interfaces to connect these cell zones together. However, you must use sliding interfaces to connect the dynamic layering regions above the valves and the remeshing region in the chamber. This is shown in Figure 10.59: The Use of Sliding Interfaces to Connect the Exhaust Valve Layering Zone to the Remeshing Zone (p. 698) with the exhaust valve almost at full extension. Notice that cells on the chamber side of the interface zone are remeshed (that is, split or merged) as the interface zone opens and closes because of the motion of the exhaust valve.

Figure 10.59: The Use of Sliding Interfaces to Connect the Exhaust Valve Layering Zone to the Remeshing Zone



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.6.4.1.3. Defining Motion/Geometry Attributes of Mesh Zones

As the piston moves down from the TDC to the BDC position, you must expand the remeshing region such that it can accommodate the valves when they are fully extended. To accomplish this, you must specify the dynamic layering zone adjacent to the piston surface to move with the piston until some specified distance from the TDC position. Beyond this cutoff distance, the motion of the layering zone is stopped and the piston wall is allowed to continue to the BDC position. Because there is relative motion between the piston head surface and the now non-moving dynamic layering zone, cell layers will be added when the ideal layer height criteria is violated. Figure 10.60: Mesh Sequence 1 (p. 699) to Figure 10.65: Mesh Sequence 6 (p. 702) show the sequence of meshes before and after the onset of cell layering when the motion in the layering zone above the piston surface is stopped (shown with $\Delta\theta$ = 5°).

Figure 10.60: Mesh Sequence 1

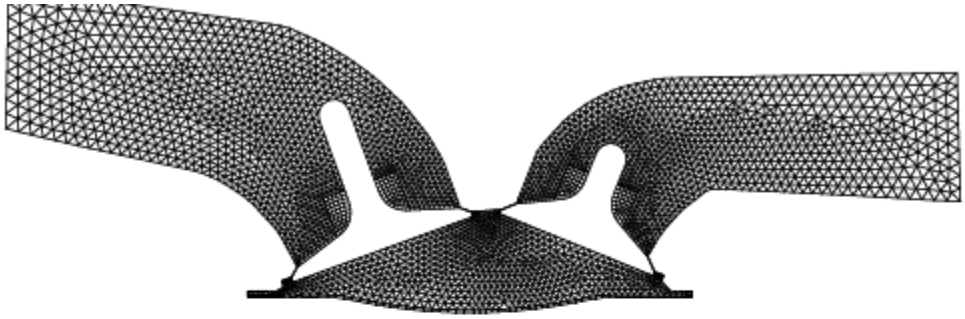


Figure 10.61: Mesh Sequence 2

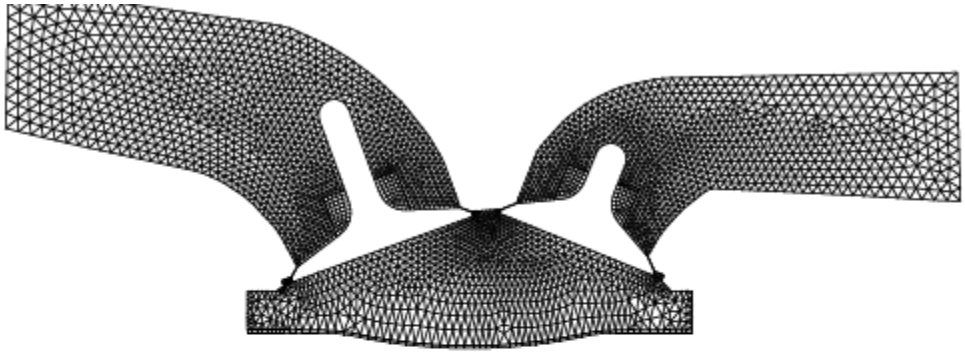
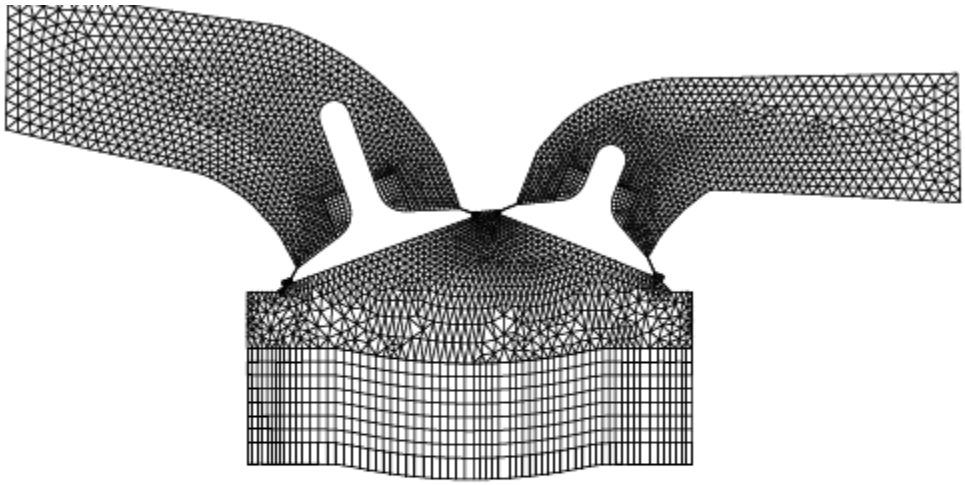


Figure 10.62: Mesh Sequence 3



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.63: Mesh Sequence 4

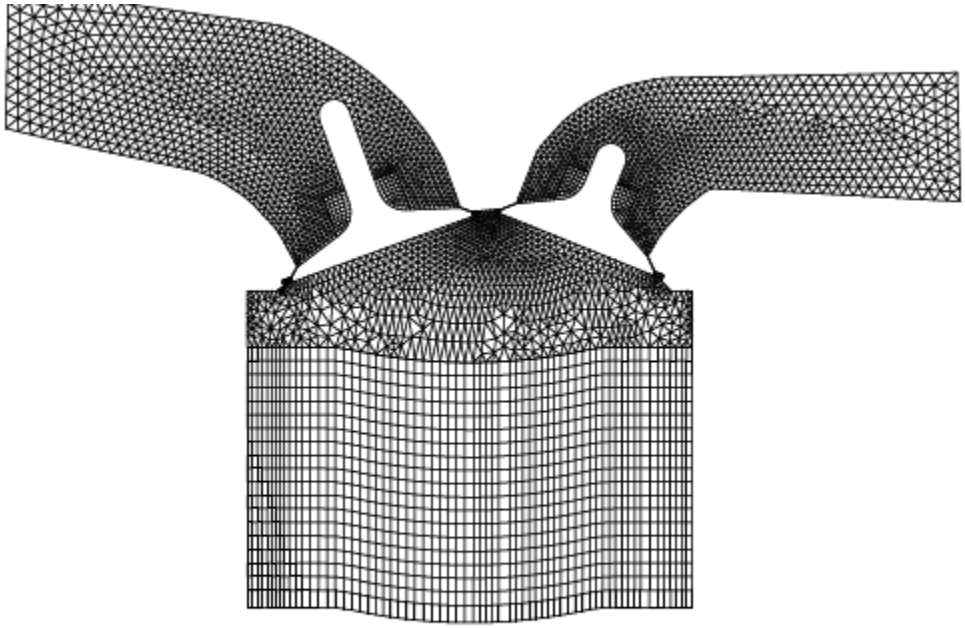
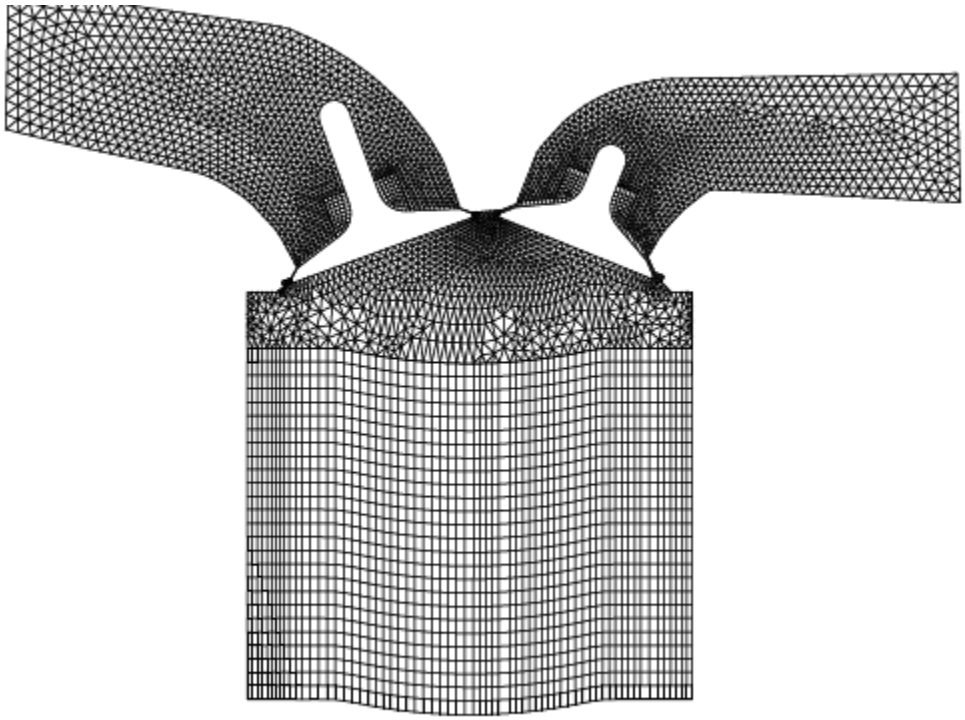
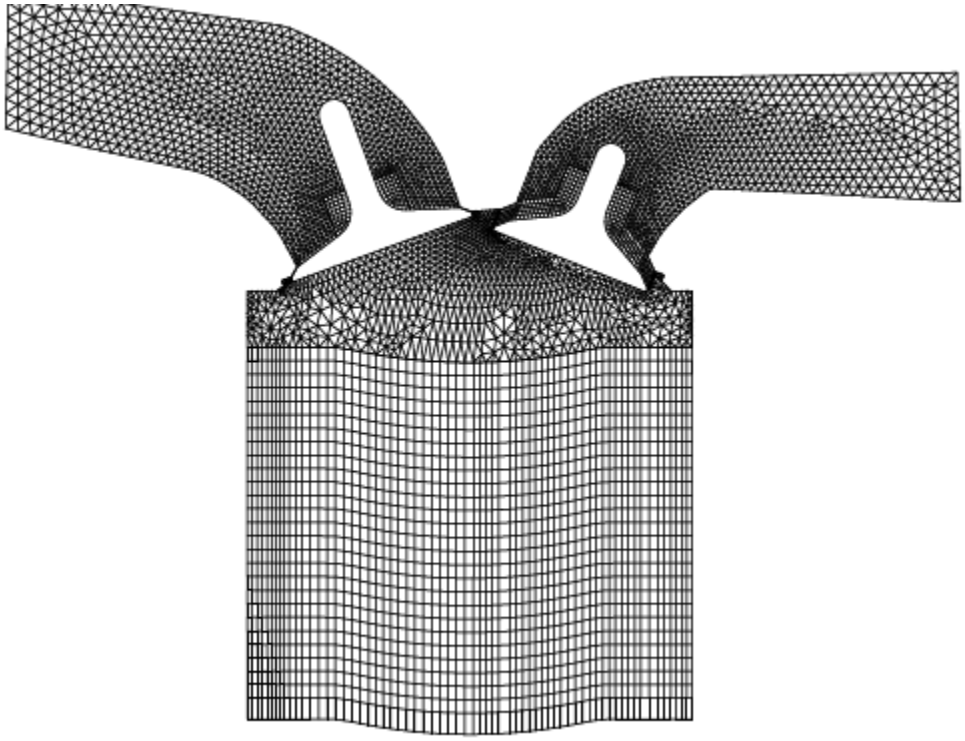


Figure 10.64: Mesh Sequence 5



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

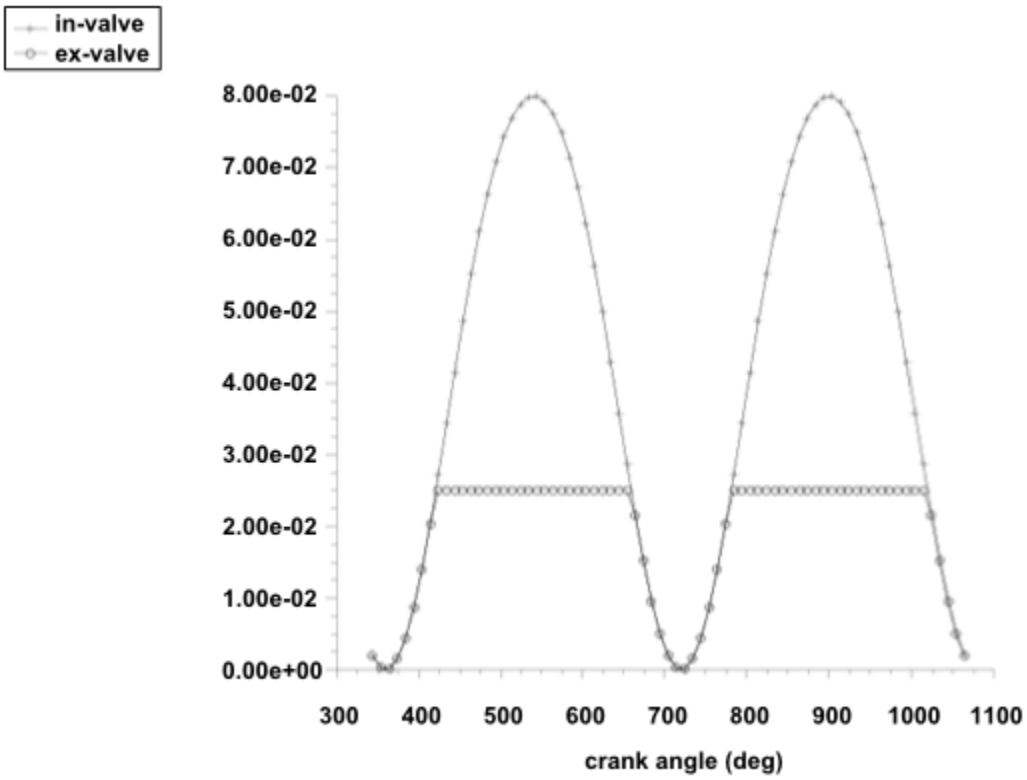
Figure 10.65: Mesh Sequence 6



ANSYS Fluent provides built-in functions to handle the full piston motion and the limited piston motion for the dynamic layering zone above the piston surface. When you define the motion attribute of the dynamic layering zone above the piston surface, you must use the limited piston motion function (****piston-limit**** in the **Motion UDF/Profile** field in the Dynamic Mesh Zones Dialog Box (p. 2490)). Note that you must define the parameters used by these functions before you can use them. In the current example, the crank radius is 40 mm and the connecting rod length is 140 mm. The piston stroke cutoff is assumed to happen at 25 mm from TDC position. The lift as a function of crank angle between 344° and 1064° is shown in Figure 10.66: Piston Position (m) as a Function of Crank Angle (deg) (p. 703) for both limited and full piston motion.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.66: Piston Position (m) as a Function of Crank Angle (deg)



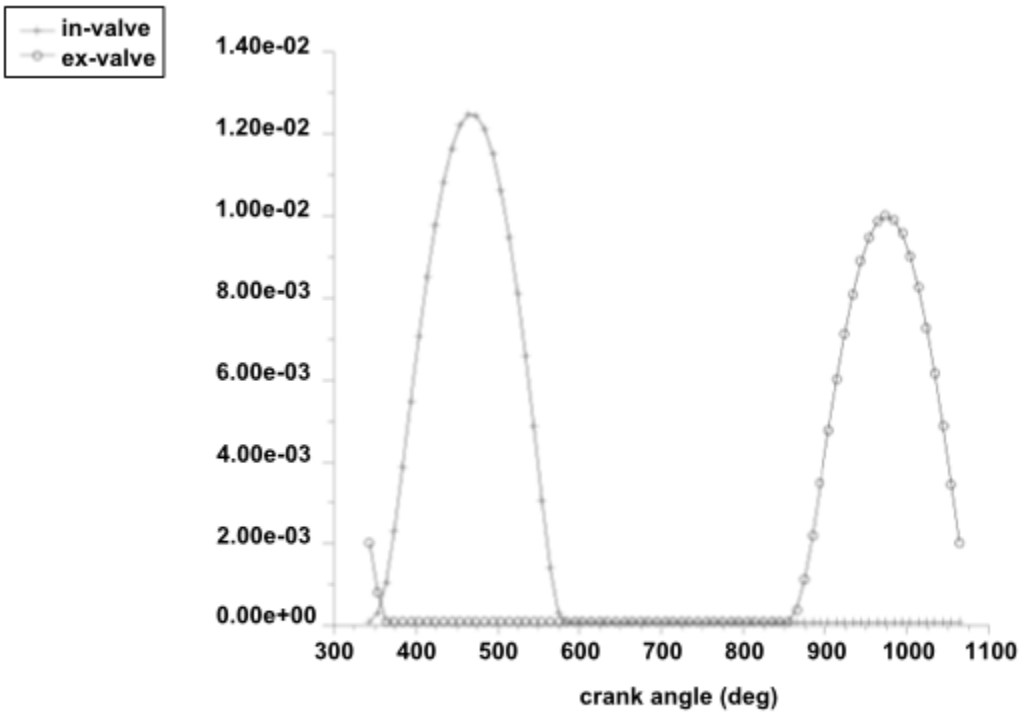
To define the motion of the valves, you must use profiles that describe the variation of valve lift with crank angle. ANSYS Fluent expects certain profile fields to be used to define the lift and the crank angle. For example, consider the following simplified profile definition:

```
((ex-valve 5 point)
(angle 0  180 270 360  720)
(lift 0.05 0.05 1.8 0.05 0.05))

((in-valve 5 point)
(angle 0  355  440 540 720)
(lift 0.05 0.05 2.0 0.05 0.05))
```

ANSYS Fluent expects the `angle` and `lift` fields to define the crank angle and lift variations, respectively. The angle must be specified in degrees and the lift values must be in meters. The actual valve lift profiles that you will use for the current example are shown in Figure 10.67: Intake and Exhaust Valve Lift (m) as a Function of Crank Angle (deg) (p. 704). Notice that there is an overlapped period where both the intake and exhaust valves are open.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.67: Intake and Exhaust Valve Lift (m) as a Function of Crank Angle (deg)



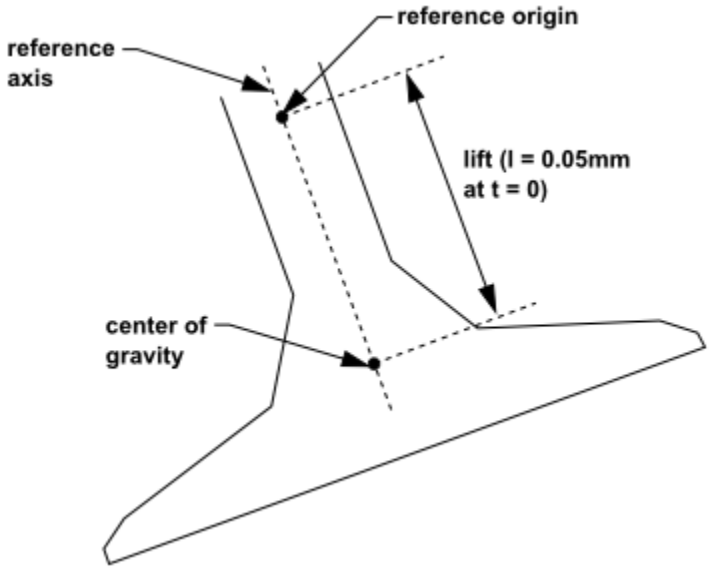
The valve lift profiles and the built-in functions will describe how each surface moves as a function of crank angle with respect to some reference point. For example, the valve lift is zero when the valve is fully closed and the valve lift is maximum when it is fully open. In order to move the surfaces, ANSYS Fluent requires that you specify the direction of motion for each surface. ANSYS Fluent will then update the "center of gravity" of each surface such that

$$\vec{x}=\vec{x}_{ref}-l\vec{e}_{axis} \tag{10.28}$$

where $\vec{x}_{ref}$ is some reference position, $\vec{e}_{axis}$ is the unit vector in the direction of motion, and l is either the valve or the piston distance with respect to the reference position $\vec{x}_{ref}$. Note that the unit vector of the direction of motion is specified to point in the negative direction. For example, the correct intake valve axis for this example is $\left(-0.3421, 0.9397\right)$, as shown in Figure 10.68: Definition of Valve Zone Attributes (Intake Valve) (p. 705).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.68: Definition of Valve Zone Attributes (Intake Valve)



10.6.4.1.4. Defining Valve Opening and Closure

ANSYS Fluent assumes that once you have set up the mesh topology, the mesh topology is unchanged throughout the entire simulation. Therefore, ANSYS Fluent does not allow you to completely close the valves such that the cells between the valve and the valve seat become degenerate (flat cells) when these surfaces come in contact (removing these flat cells would require the creation of new boundary face zones). To prevent the collapse, you must define a minimum valve lift and ANSYS Fluent will automatically stop the motion of the valve when the valve lift is smaller than the minimum valve lift value. The minimum valve lift value can be specified in the **In-Cylinder** tab of the Options Dialog Box (p. 2478). For the current example, a minimum valve lift value of 0.1 mm is assumed.

When the valve position is smaller than the minimum valve lift value, it is normal practice to assume that the valve is closed. The actual closing of the valves is accomplished by deleting the sliding interfaces that connect the chamber cell zone to the dynamic layering zones on the valves. The interface zones are then converted to walls to close off the "gaps" between the valves and the valve seats.

The valve opening is achieved by the reverse process. When the valve lift has reached beyond the minimum valve lift value, the valve is assumed to be open and you can redefine the sliding interfaces such that the chamber zone is now connected to the dynamic layering zones above the valves.

10.6.5. Six DOF Solver Settings

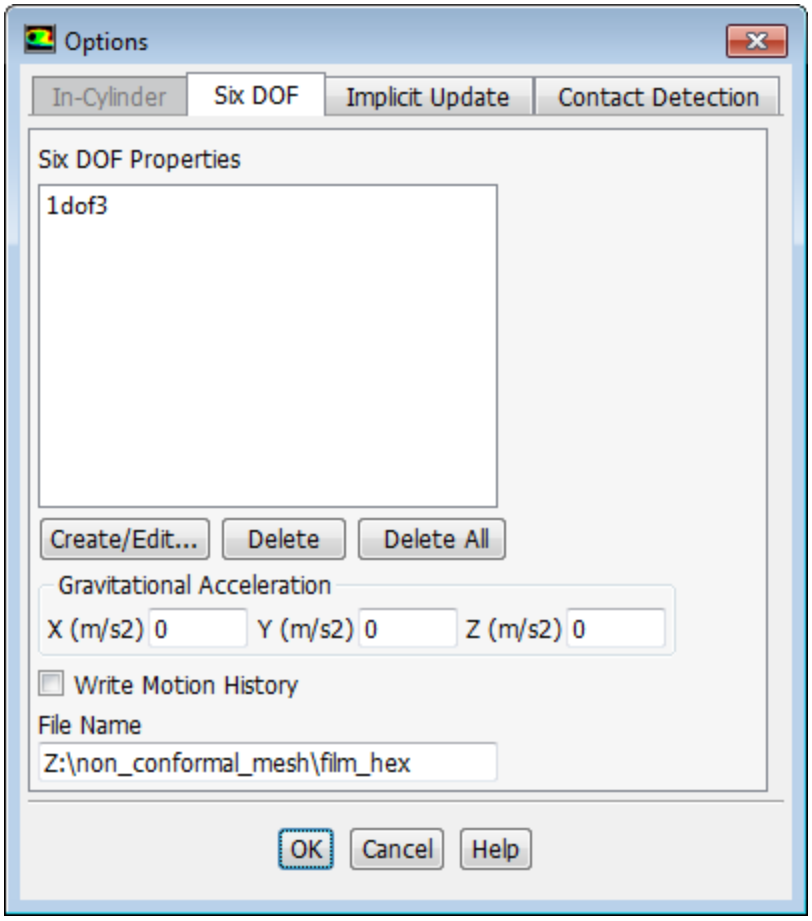
ANSYS Fluent's six degree of freedom (six DOF) solver computes external forces and moments (such as aerodynamic and gravitational forces and moments) on objects that undergo rigid body motion. These forces are computed by numerical integration of pressure and shear stress over the object's surfaces. Additional load forces can be added, such as injector forces, thrust (propulsive) forces, and moments produced by a coil spring. This technique, along with the ANSYS Fluent solver and the use of dynamic meshes, can be readily applied to many useful applications, such as store separation [102] (p. 2892), [111] (p. 2893). Note that the objects may be comprised of multiple zones.

To use the six degree of freedom solver for your transient dynamic mesh simulation, perform the following steps:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Enable the **Six DOF** option in the **Options** group box of the Dynamic Mesh Task Page (p. 2471) (Figure 10.13: The Dynamic Mesh Task Page (p. 643)) and click the **Settings...** button. The **Options** dialog box will open, where you can click the **Six DOF** tab (Figure 10.69: The Six DOF Tab of the Options Dialog Box (p. 706)).

   Figure 10.69: The Six DOF Tab of the Options Dialog Box



   This tab allows you to perform the following actions:

   - You can create and manage sets of six DOF properties for rigid body motion by using the list and buttons under **Six DOF Properties**; otherwise, you will have to create one or more user-defined functions. See Setting Rigid Body Motion Attributes for the Six DOF Solver (p. 707) for details.
   - You can specify the gravitational acceleration for the *x*, *y*, and *z* directions either in this dialog box, or in the **Operating Conditions** dialog box.
   - You can keep track of an object's motion history by enabling the **Write Motion History** option and entering a **File Name**. A single motion history file will be generated for each moving object, which can be used to display zone motion for postprocessing your results.

2. If the mesh motion of your six DOF simulation depends on the fluid flow, it is beneficial to enable implicit mesh updating (as described in Implicit Update Settings (p. 709)). If the implicit mesh update is not sufficient to stabilize the solver due to the fluid motion, solution stabilization may be necessary. The solution stabilization settings are set in the **Solver Options** tab of the **Dynamic Mesh Zones** dialog box for six DOF zones (Rigid Body Motion (p. 726)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Define the rigid-body zone(s) that make up each moving object by using the Dynamic Mesh Zones Dialog Box (p. 2490), as described in Rigid Body Motion (p. 726).

Setup → Dynamic Mesh → Create/Edit...

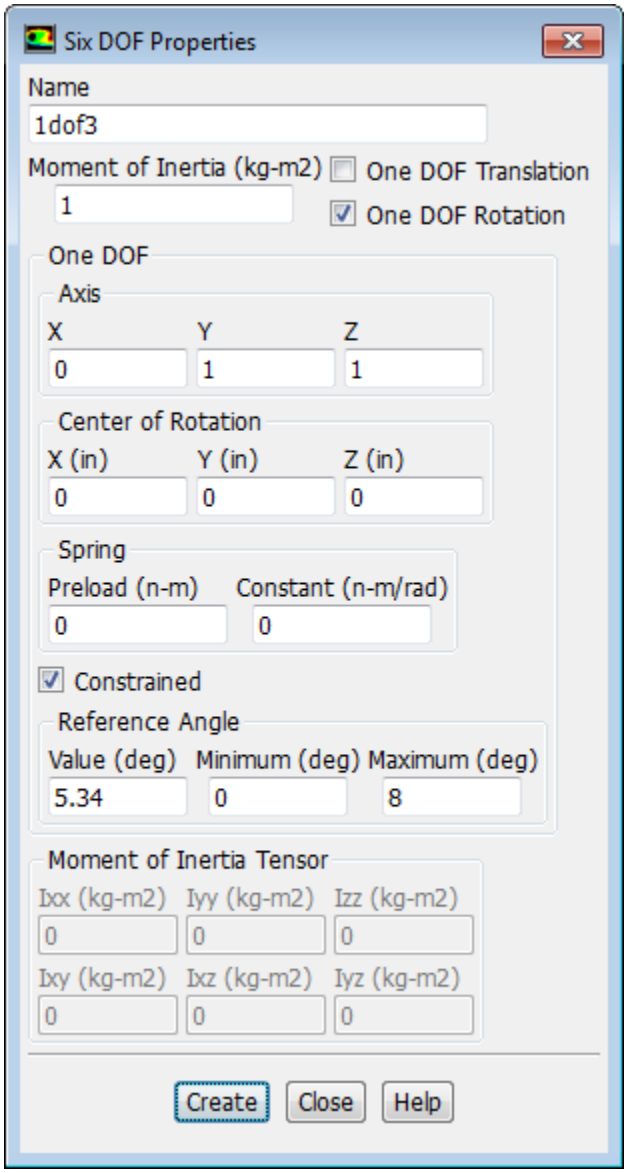
10.6.5.1. Setting Rigid Body Motion Attributes for the Six DOF Solver

As part of the six DOF solver settings, you must provide either a set of properties or a user-defined function (UDF) for each moving object, in order to define its overall mass, rotational inertia, constraints, loading from sources other than aerodynamics and gravity, and so on. Each object needs exactly one set of properties / UDF, regardless of how many zones make up the object. Note that if you have multiple identical objects, you can create an initial set of properties / UDF, and then make copies with unique names.

If the motion of the object is complex or if only half of the object is modeled using a symmetry zone, then you will need to use a UDF (see `DEFINE_SDOF_PROPERTIES` in the *Fluent Customization Manual* for the details); otherwise, it is easier to use a set of properties. Such sets are created using the **Six DOF Properties** dialog box, which is opened by clicking the **Create/Edit...** button in the **Six DOF** tab of the Options Dialog Box (p. 2478).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.70: The Six DOF Properties Dialog Box



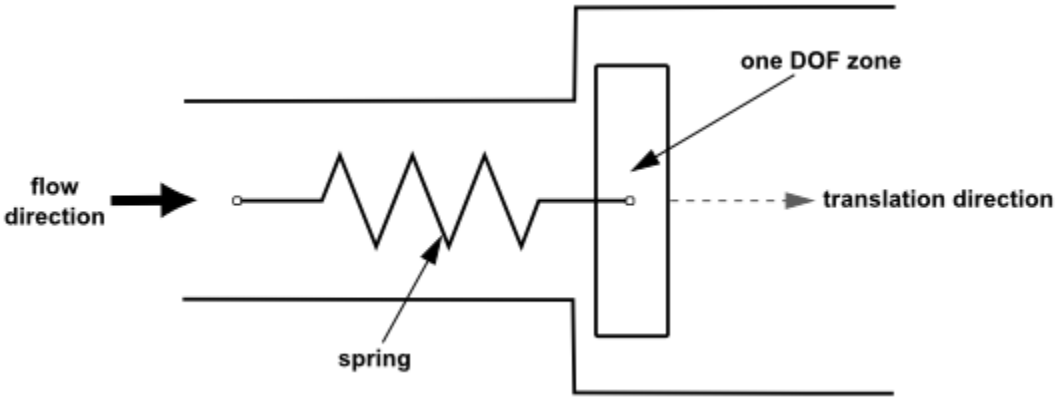
In the **Six DOF Properties** dialog box, you will need to provide a **Name** for the set of properties, define the other fields in the dialog box, and click **Create**.

For six DOF motion, the only other fields that must be defined are the **Mass** and the **Moments of Inertia Tensor** components. Note that the inertia tensor components are defined relative to the **Center of Gravity** and **Rigid Body Orientation** settings defined in the Dynamic Mesh Zones Dialog Box (p. 2490); the latter settings allow you to use a coordinate system that is local (rather than global), if that is more convenient.

If you want to specify that the dynamic object be limited to one DOF motion (that is, either a simple translation or a simple rotation), enable the **One DOF Translation** or **One DOF Rotation** option and define the settings in the **One DOF** group box. The following figure shows an example of a one DOF translation:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.71: A Check Valve with One DOF Translation



Note the following about one DOF motion settings:

- For rotation, you will define the **Moment of Inertia** for the dynamic object, rather than the **Mass**.
- For 2D cases with rotation, the axis will be either the positive or negative *z* direction, depending on the sign of the value you enter for **Z** in the **Axis** group box.
- You have the option of defining a Hooke's law **Spring** that exerts forces / torques on the dynamic object. It is a compression / extension spring for translations, and it is a torsion spring for rotations.

  For example: for a translation, the specified **Constant** will be multiplied by the distance of the dynamic object from its initial location. This force will be combined with a **Preload** value that is constantly applied to the dynamic object, in order to produce the net spring loading. Note that the **Preload** can be a positive or negative value, and is defined relative to the **Direction**.
- You can limit the range of motion of the dynamic object by enabling the **Constrained** option and defining the settings in the **Reference Point** group box (for translation) or the **Reference Angle** group box (for rotation).

  For example: for a translation, you could enter 2 inches for the **Location**, in order to assign a coordinate value to a point located on the dynamic object in its initial position. Then you would define the range of motion allowed relative to that coordinate: you would enter `1.2` and 2 inches for the **Minimum** and **Maximum**, respectively, if you wanted the object to only be allowed to move .8 inches in the direction opposite to the **Direction** vector. The value of the **Location** is arbitrary, and allows you to define the range using values that are most convenient for your problem.
- The initial position / orientation of the object (which is used to calculate the spring loading and/or range of motion) is established when you create the dynamic zone(s) using the Dynamic Mesh Zones Dialog Box (p. 2490).

10.6.6. Implicit Update Settings

For transient problems, you can enable implicit mesh updating when you want to have the dynamic mesh updated during a time step (as opposed to just at the beginning of a time step). This capability is beneficial only for applications in which the mesh motion depends on the flow field (for example, cases that use the six DOF solver or involve fluid-structure interaction). For such applications, having the mesh motion updated within the time step based on the converging flow solution results in a stronger coupling between the flow solution and the mesh motion, and leads to a more robust solver

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

run. Implicit mesh updating allows you to run simulations that otherwise could not be solved or would require an unreasonably small time step.

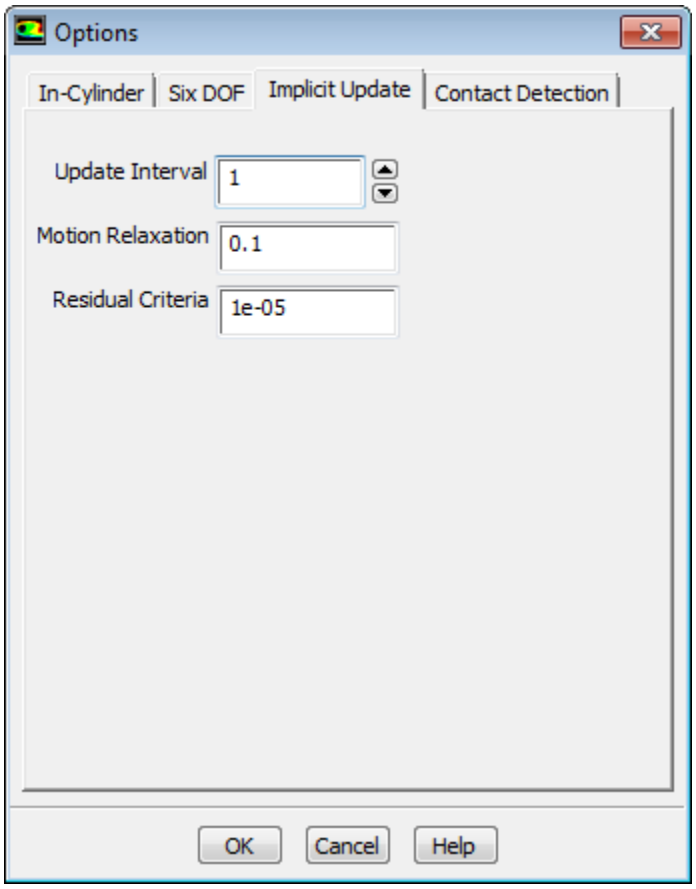
Note that implicit mesh updating cannot be used with the following:

- the density-based solver when **Explicit** is selected from the **Transient Formulation** drop-down menu in the **Solution Methods** task page
- steady-state solutions
- in-cylinder applications

To enable implicit mesh updating, perform the following steps:

1. Enable **Implicit Update** in the **Options** group box of the Dynamic Mesh Task Page (p. 2471) (Figure 10.13: The Dynamic Mesh Task Page (p. 643)).
2. Click the **Settings...** button to open the Options Dialog Box (p. 2478).

   Figure 10.72: The Implicit Update Tab of the Options Dialog Box



   Click the **Implicit Update** tab and enter values for the settings.

   a. Enter a value for **Update Interval**, in order to specify the frequency in iterations at which the mesh will be updated within a time step.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Enter a value (within the range of 0 to 1) for **Motion Relaxation**, in order to define the relaxation of the motion (that is, displacement of the nodes) during the mesh update. The relaxation of the displacements is defined by the following equation:

$$x_k = \omega(x_{computed,k}) + (1-\omega)x_{k-1} \tag{10.29}$$

where x_k is the node position at iteration k (within a time step), $x_{computed,k}$ is the computed node position (based on the flow field), and ω is the motion relaxation.

c. Enter a value for **Residual Criteria**, in order to set the relative residual threshold that is used to check the motion convergence. The residual criteria is applied to a relative residual. ANSYS Fluent scales the difference between the motion in iteration k and iteration $k-1$ by the motion computed at the beginning of the time step. If this relative motion difference is smaller than the residual criteria, the mesh motion is considered converged.

3. If you are using a UDF to compute the motion, make sure that the UDF uses the current flow field during each call to compute the motion (that is, no previously stored information should be used). This is necessary, as the UDF will be called each time the mesh is updated — which can be several times within a time step, depending on what you entered for the **Update Interval**.

4. After you run the simulation, make sure that the motion (and consequently, the solution) is properly converged. If the motion requires more iterations to converge than the flow field, a warning will be displayed in the console during the iteration process. Note that the maximum number of iterations per time step (defined in the **Run Calculation** task page) is respected by the mesh motion convergence check.

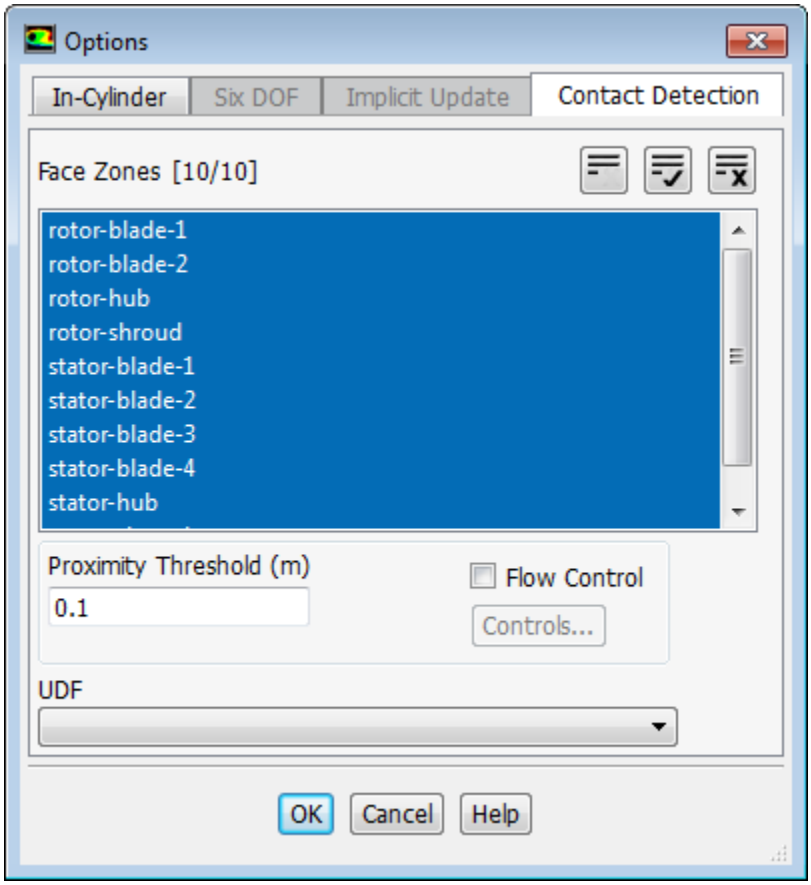
10.6.7. Contact Detection Settings

Contact detection is used to detect if the computed mesh motion will result in contact of a moving surface with other surrounding surfaces. If there is contact within specified tolerances, then the mesh motion of the moving zone can be constrained using nodal contact information within user-defined functions (See Example 2 under DEFINE_CONTACT in the Fluent).

To enable contact detection in your dynamic mesh simulation, select the **Contact Detection** check box in the **Options** group of the **Dynamic Mesh** task page. Click the **Settings...** button to open the **Options** dialog box and select the **Contact Detection** tab (see Figure 10.73: The Contact Detection Tab of the Options Dialog Box (p. 712)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.73: The Contact Detection Tab of the Options Dialog Box



You can select the face zones that will be involved in the contact detection process from the **Face Zones** list. While all eligible face zones in the **Face Zones** list are selected by default, you can choose to exclude certain zones for efficiency. Only walls and dynamic zones of type **Rigid Body** or **User-Defined** can be selected for contact detection.

From the **UDF** drop-down list, you can specify a user-defined function that will be invoked when contact has been detected. For more information about contact UDFs, see `DEFINE_CONTACT` in the Fluent Customization Manual.

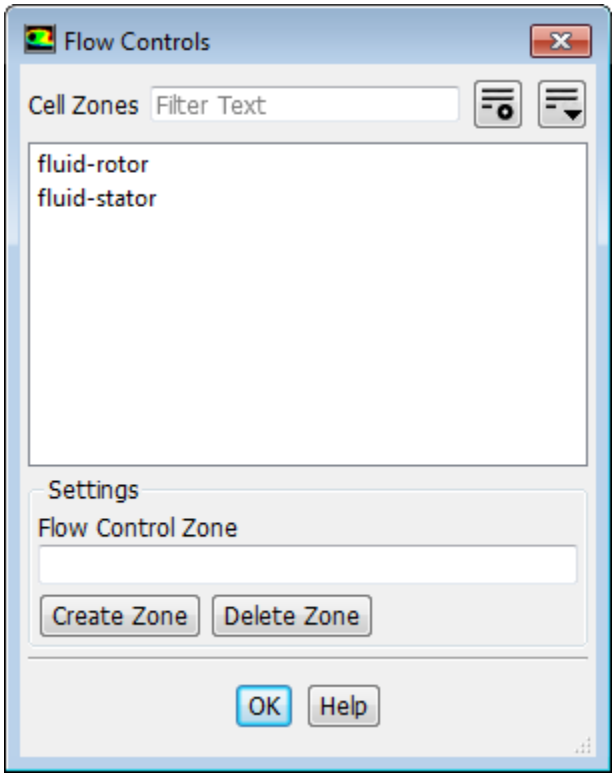
You must specify a **Proximity Threshold** value in order to enable the contact detection process. When the distance between face zones falls below this threshold value, the UDF specified in the **UDF** drop-down list is invoked.

Cells adjacent to face zones that fall below the **Proximity Threshold** distance can also be tagged and separated into new cell zones during the contact detection process. This is useful for specifying different flow conditions in contact regions during the simulation.

To enable flow control options, select the **Flow Control** check box in the **Contact Detection** tab. Additional settings for flow control can then be specified by clicking the **Controls...** button to display the **Flow Controls** dialog box (see Figure 10.74: The Flow Controls Dialog Box (p. 713)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.74: The Flow Controls Dialog Box



In the **Flow Controls** dialog box, you can specify a flow control zone (for example, restrictions, and so on) for any cell zone in the mesh. To create a flow control zone, select a zone from the **Cell Zones** list, specify a new name in the **Flow Control Zone** text box, and click the **Create Zone** button. Once the flow control zone has been created, it becomes available in the **Cell Zone Conditions** task page, where additional physical properties (for example, inertial and/or viscous porous resistance) can be specified. During the simulation, once cells in the contact region have been separated into new zones, the physical properties specified in the respective flow control zone are copied to the newly separated cell zone.

10.6.8. Defining Dynamic Mesh Events

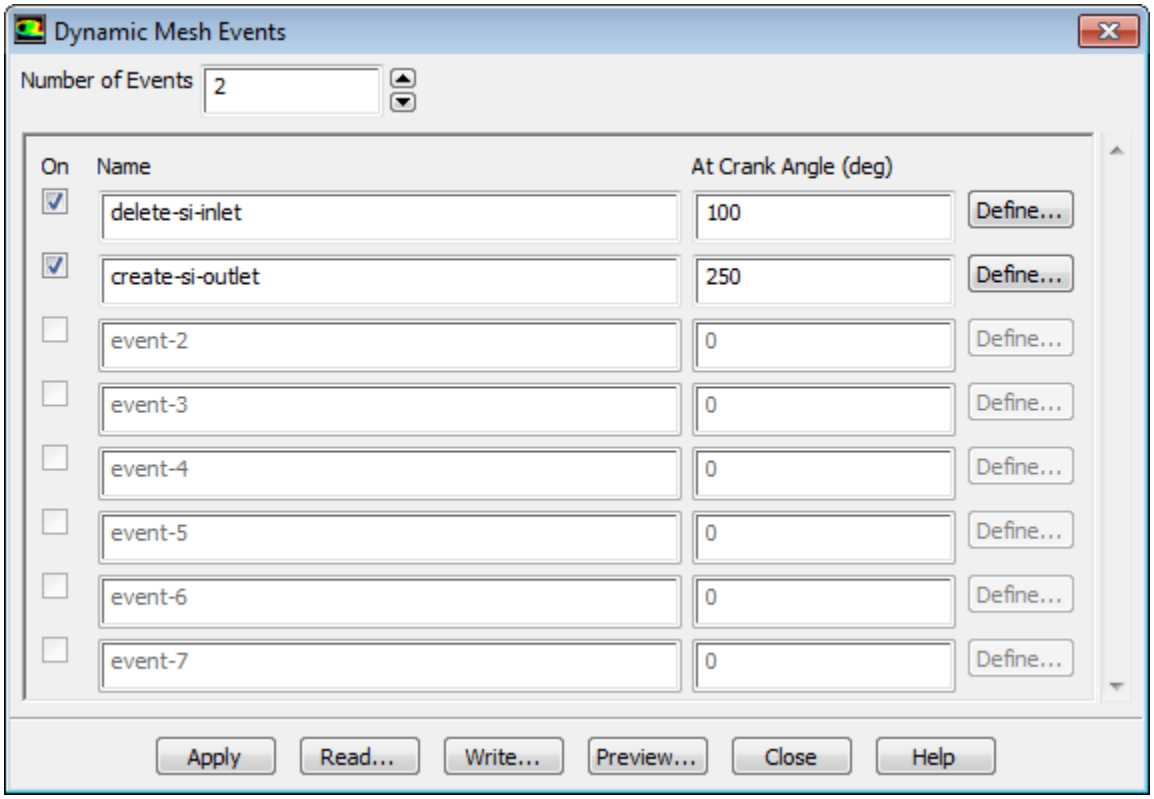
If you are simulating a transient flow, you can use the events in ANSYS Fluent to control the timing of specific events during the course of the simulation. With in-cylinder flows for example, you may want to open the exhaust valve (represented by a pair of deforming sliding interfaces) by creating an event to create the sliding interfaces at some crank angle. You can also use dynamic mesh events to control when to suspend the motion of a face or cell zone by creating the appropriate events based on the crank angle or time. Note that in-cylinder flows are crank angle-based, whereas all other flows are time-based.

10.6.8.1. Procedure for Defining Events

You can define the events using the Dynamic Mesh Events Dialog Box (p. 2486) (Figure 10.75: The Dynamic Mesh Events Dialog Box (p. 714)).

Setup → Dynamic Mesh → Events...

Figure 10.75: The Dynamic Mesh Events Dialog Box



The procedure for defining events is as follows:

1. Increase the **Number of Events** value to the number of events you want to specify. As this value is increased, additional event entries in the dialog box will become editable.

2. Enable the check box next to the first event and enter a name for the event under the **Name** heading.

3. Specify either the time or the crank angle at which you want the event to occur.

   For in-cylinder flows, specify the crank angle at which you want the event to occur under **At Crank Angle**.

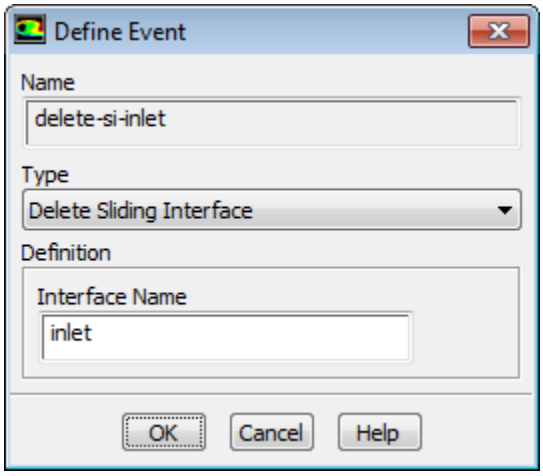
   For non-in-cylinder flows, specify the time (in seconds) at which you want the event to occur under **At Time**.

   It is not necessary to specify the events in order of increasing time or crank angle, but it may be easier to keep track of events if you specify them in the order of increasing time or angle.

4. Click the **Define...** button to open the Define Event Dialog Box (p. 2487) (Figure 10.76: The Define Event Dialog Box (p. 715)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.76: The Define Event Dialog Box



5. In the Define Event Dialog Box (p. 2487), choose the type of event by selecting **Change Zone Type**, **Copy Zone BC**, **Activate Cell Zone**, **Deactivate Cell Zone**, **Create Sliding Interface**, **Delete Sliding Interface**, **Change Motion Attribute**, **Change Time Step Size**, **Change Under-Relaxation Factors**, **Insert Boundary Zone Layer**, **Remove Boundary Zone Layer**, **Insert Interior Zone Layer**, **Remove Interior Zone Layer**, **Insert Cell Layer**, **Remove Cell Layer**, **Execute Command**, **Replace Mesh**, **Inert EGR Reset**, or **Diesel Unsteady Flamelet Reset** in the **Type** drop-down list. These event types and their definitions are described later in this section.

6. Repeat steps 2–5 for the other events, if relevant.

7. Click **Apply** in the Dynamic Mesh Events Dialog Box (p. 2486) after you finish defining all events.

8. To play the events to check that they are defined correctly, click the **Preview...** button in the Dynamic Mesh Events Dialog Box (p. 2486). This displays the Events Preview Dialog Box (p. 2489).

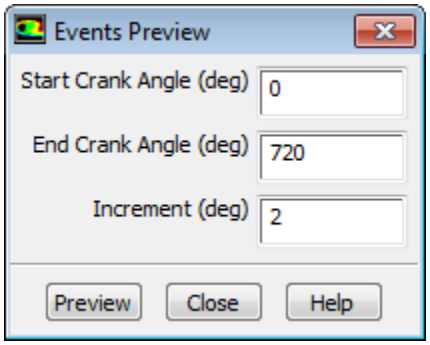
   For in-cylinder flows, you use the Events Preview Dialog Box (p. 2489) (Figure 10.77: The Events Preview Dialog Box for In-Cylinder Flows (p. 716)), to enter the crank angles at which you want to start and end the playback in the **Start Crank Angle** and **End Crank Angle** fields, respectively.

   For non-in-cylinder flows, you use the Events Preview Dialog Box (p. 2489) to enter the time at which you want to start and end the playback in the **Start Time** and **End Time** fields, respectively.

   Specify the size of the step to take during the playback in the **Increment** field. Click **Preview** to play back the events. ANSYS Fluent will play the events at the time (or crank angle in the case of in-cylinder flows) specified for each event and report when each event occurs in the text (console) window.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.77: The Events Preview Dialog Box for In-Cylinder Flows



For in-cylinder simulations, you must specify the events for one complete engine cycle. In the subsequent cycles, the events are executed whenever

$$\theta_{event} = \theta_c \pm n\theta_{period} \tag{10.30}$$

where θ_{event} is the event crank angle, θ_c is the current crank angle calculated from Equation 10.26 (p. 692), θ_{period} is the crank angle period for one cycle, and n is some integer.

As an example, for in-cylinder simulations, you are not required to specify the event crank angle to correspond exactly to the current crank angle calculated from Equation 10.26 (p. 692). ANSYS Fluent will execute an event if the current crank angle is between $\pm 0.5\Delta\theta$ where $\Delta\theta$ is the equivalent change in crank angle for the time step. For example, if the event preview is executed between crank angle of 340° and 1060° (crank period is 720°) using an increment of 1°, ANSYS Fluent will report the following in the text window.

```
Execute Event: open-in-valve-left (defined at: 353.10, current angle: 353.00)
Execute Event: open-in-valve-right (defined at: 353.00, current angle: 353.00)
Execute Event: close-ex-valve-right (defined at: 355.60, current angle: 356.00)
Execute Event: close-ex-valve-left (defined at: 357.80, current angle: 358.00)
Execute Event: close-in-valve-left (defined at: 571.60, current angle: 572.00)
Execute Event: close-in-valve-right (defined at: 571.80, current angle: 572.00)
Execute Event: open-ex-valve-right (defined at: 137.10, current angle: 857.00)
Execute Event: open-ex-valve-left (defined at: 139.00, current angle: 859.00)
```

Notice that events defined at 137.10° and 139° are executed at 857° and 859°, respectively, because they satisfy the condition of Equation 10.30 (p. 716).

10.6.8.2. Defining Events for In-Cylinder Applications

ANSYS Fluent will automatically limit the valve lift values depending on the specified minimum valve lift value. However, the conversion of the sliding interface zones to walls (and vice versa) is accomplished via the in-cylinder events (see Defining Dynamic Mesh Events (p. 713)). For example, if the exhaust valve closes at -5° before TDC position, you must define a **Delete Sliding Interface** event at the crank angle of -5°. You must define similar events for the intake valve opening (using the **Create Sliding Interface** event), the intake valve closing (**Delete Sliding Interface** event), and the exhaust valve opening (**Create Sliding Interface** event) at the respective crank angles.

For the current example, the exhaust valve is assumed to be open between 131° and 371° and the intake valve is open between at 345° and 584°.

10.6.8.2.1. Events

Each of the available events is described below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.6.8.2.2. Changing the Zone Type

You can change the type of a zone to be a wall, or an interface, interior, fluid, or solid zone during your simulation. To change the type of a zone, select **Change Zone Type** in the **Type** drop-down list in the Define Event Dialog Box (p. 2487) (Figure 10.76: The Define Event Dialog Box (p. 715)). Select the zone(s) that you want to change in the **Zone** list, and then select the new zone type in the **New Zone Type** drop-down list.

10.6.8.2.3. Copying Zone Boundary Conditions

You can copy boundary conditions from one zone to other zones during your simulation. If, for example, you have changed an inlet zone to type wall with the **Change Zone Type** event, you can set the boundary conditions of the new zone type by simply copying the boundary conditions from a known zone with the corresponding zone type.

To copy boundary conditions from one zone to another, select **Copy Zone BC** in the **Type** drop-down list in the Define Event Dialog Box (p. 2487) (Figure 10.76: The Define Event Dialog Box (p. 715)). In the **From Zone** drop-down list, select the zone that has the conditions you want to copy. In the **To Zone(s)** list, select the zone or zones to which you want to copy the conditions.

ANSYS Fluent will set *all* of the boundary conditions for the zones selected in the **To Zone(s)** list to be the same as the conditions for the zone selected in the **From Zone** list. (You cannot copy a subset of the conditions, such as only the thermal conditions.)

Note that you cannot copy conditions from external walls to internal (that is, two-sided) walls, or vice versa, if the energy equation is being solved, since the thermal conditions for external and internal walls are different.

10.6.8.2.4. Activating a Cell Zone

To activate a cell zone, select **Activate Cell Zone** in the **Type** drop-down list in the Define Event Dialog Box (p. 2487) (Figure 10.76: The Define Event Dialog Box (p. 715)), then select the zone that you want to activate in the **Zone(s)** list. For more information, see Replacing, Deleting, Deactivating, and Activating Zones (p. 218).

10.6.8.2.5. Deactivating a Cell Zone

To deactivate a cell zone, select **Deactivate Cell Zone** in the **Type** drop-down list in the Define Event Dialog Box (p. 2487) (Figure 10.76: The Define Event Dialog Box (p. 715)), then select the zone that you want to deactivate in the **Zone(s)** list.

Only deactivated zones can be activated. When a zone is deactivated, ANSYS Fluent skips the zone during the calculations. For more information, see Replacing, Deleting, Deactivating, and Activating Zones (p. 218).

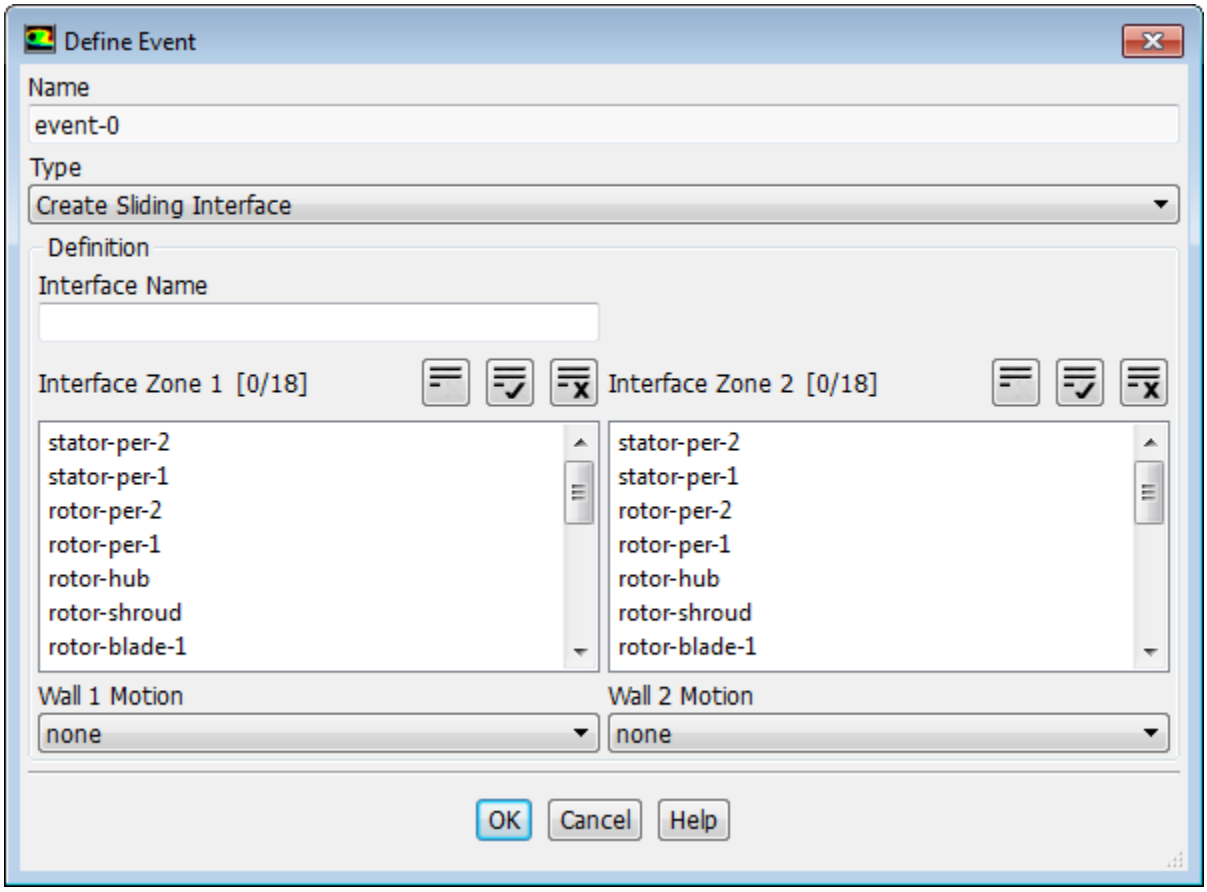
10.6.8.2.6. Creating a Sliding Interface

To create a sliding interface during your simulation, select **Create Sliding Interface** in the **Type** drop-down list in the Define Event Dialog Box (p. 2487) (Figure 10.78: The Define Event Dialog Box for the Creating Sliding Interface Option (p. 718)). Enter a name for the sliding interface in the **Interface Name** field. Select the zones on either side of the interface in the **Interface Zone 1** and **Interface Zone 2** drop-down lists.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You have the option to select any number of zones listed under each of the interface zones. ANSYS Fluent calculates intersections between all possible combinations of the left and right side of the interfaces, allowing you more flexibility in terms of creating zones and defining the interfaces.

Figure 10.78: The Define Event Dialog Box for the Creating Sliding Interface Option



Important

If ANSYS Fluent finds another interface with the same name as defined in the event, then the old interface will be deleted and a new one created as defined in the dynamic mesh event.

If the interface zones that you selected above do not overlap each other completely, the non-overlapped regions on each interface zones are put into separate wall zones by ANSYS Fluent. If these wall zones (that is, non-overlapped regions) have motion attributes associated with them, their motion can only be specified by copying the motion from another dynamic zone by selecting the appropriate dynamic zones in the **Wall 1 Motion** and **Wall 2 Motion** drop-down lists, respectively.

Note that you do not have to change the boundary type from wall to interface. When the **Create Sliding Interface** event is executed, ANSYS Fluent will automatically change the boundary type of the face zones selected in **Interface Zone 1** and **Interface Zone 2** to type interface before the sliding interface is created.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.6.8.2.7. Deleting a Sliding Interface

To delete a sliding interface that has been created earlier in your simulation, select **Delete Sliding Interface** in the **Type** drop-down list in the Define Event Dialog Box (p. 2487) (Figure 10.76: The Define Event Dialog Box (p. 715)). Enter the name of the sliding interface to be deleted in the **Interface Name** field.

As with the **Create Sliding Interface** event, ANSYS Fluent will automatically change the corresponding interface zones to wall. However, you may want to use the **Copy Zone BC** event to set any boundary conditions that are not the default conditions that ANSYS Fluent assumes.

10.6.8.2.8. Changing the Motion Attribute of a Dynamic Zone

To change the motion attribute of a dynamic zone during your in-cylinder calculation, select **Change Motion Attribute** in the **Type** drop-down list in the Define Event Dialog Box (p. 2487) (Figure 10.76: The Define Event Dialog Box (p. 715)). Select the **Attribute** (**smoothing** or **remeshing**) and set the appropriate **Status** (**enable** or **disable**). Select the corresponding dynamic zones for which you want to change the motion attributes in the **Dynamic Zones** list.

The **smoothing** attribute is used to enable or disable smoothing of nodes on selected deforming face zones and the **remeshing** attribute is used to enable and disable face remeshing on selected deforming face zones.

10.6.8.2.9. Changing the Time Step

To change the time step at some point during the simulation, select **Change Time Step Size** in the **Type** drop-down list in the Define Event Dialog Box (p. 2487). Specify the new physical time step size by entering the new **Time Step Size** in seconds.

For in-cylinder simulations, specify the new physical time step by entering the new **Crank Angle Step Size** value in degrees. The physical time step is calculated from

$$\Delta t = \frac{\Delta \theta_c}{\Omega_{shaft}} \tag{10.31}$$

10.6.8.2.10. Changing the Under-Relaxation Factor

To change one or more under-relaxation factors, select **Change Under-Relaxation Factor** in the **Type** drop-down list in the Define Event Dialog Box (p. 2487) (Figure 10.76: The Define Event Dialog Box (p. 715)). Select the under-relaxation factor that you want to change, and assign a new value to it in the **Under-Relaxation Factors** list. For more information on setting under-relaxation factors, see Setting Under-Relaxation Factors (p. 1564).

10.6.8.2.11. Inserting a Boundary Zone Layer

To insert a new cell zone layer as a separate cell zone adjacent to a boundary, select **Insert Boundary Zone Layer** in the **Type** drop-down list in the Define Event Dialog Box (p. 2487). Specify the **Base Dynamic Zone**, from which the layer of cells is to be created, and the **Side Dynamic Zone**, which represents the deforming face zone adjacent to the **Base Dynamic Zone** before the layer is inserted. The new cell zone will inherit the boundary conditions of the cell zone adjacent to the **Base Dynamic Zone** before the layer is inserted.

Note that a new cell layer can be inserted only from a one-sided **Base Dynamic Zone**. You cannot insert a new cell layer from an interior face zone.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.79: Boundary Zone Before Insertion (p. 720) and Figure 10.80: Boundary Zone After Insertion (p. 720) illustrate the insertion of a boundary zone layer. In both figures, the circular face at the top of the cylinder is the base dynamic zone.

Figure 10.79: Boundary Zone Before Insertion

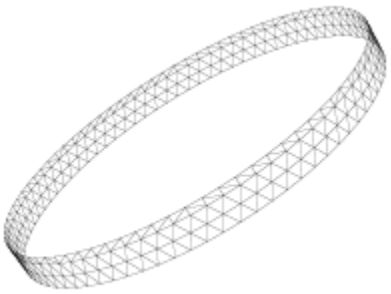


Figure 10.80: Boundary Zone After Insertion

10.6.8.2.12. Removing a Boundary Zone Layer

To remove the cell zone layer inserted using the **Insert Boundary Zone Layer** event, select **Remove Boundary Zone Layer** in the **Type** drop-down list in the Define Event Dialog Box (p. 2487). Specify the same **Base Dynamic Zone** that you used when you defined the insert boundary layer event.

Note that a cell layer can be removed only from a one-sided **Base Dynamic Zone**.

10.6.8.2.13. Inserting an Interior Zone Layer

To insert a new zone layer as a separate cell zone adjacent to the internal side of a boundary, select **Insert Interior Zone Layer** in the **Type** drop-down list in the Define Event Dialog Box (p. 2487). Specify the **Base Dynamic Zone** and the **Side Dynamic Zone** as described in the **Insert Boundary Zone Layer** event. You also must specify the names of the new interior face zones (**Internal Zone 1 Name** and **Internal Zone 2 Name**) that will be created after the cell zone layer is created by ANSYS Fluent.

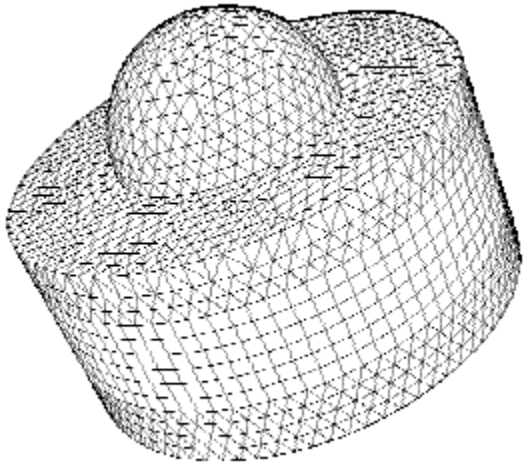
ANSYS Fluent inserts the interior cell layer by splitting the cell zone adjacent to the **Base Dynamic Zone** with a plane. The position of the plane and the normal direction of the plane are implicitly defined by the cylinder origin and cylinder axis of the **Side Dynamic Zone**.

Figure 10.81: Interior Zone Before Insertion (p. 721) and Figure 10.82: Interior Zone After Insertion (p. 721) illustrate the insertion of an interior zone layer.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.81: Interior Zone Before Insertion

Figure 10.82: Interior Zone After Insertion



10.6.8.2.14. Removing an Interior Zone Layer

To remove the zone layer inserted using the **Insert Interior Zone Layer** event, select **Remove Interior Zone Layer** in the **Type** drop-down list in the Define Event Dialog Box (p. 2487). Specify the same **Internal Zone 1 Name** and **Internal Zone 2 Name** that you used to define the **Insert Interior Zone Layer** event.

10.6.8.2.15. Inserting a Cell Layer

To manually insert a new cell layer to the existing cell zone, select **Insert Cell Layer** in the **Type** drop-down list in the Define Event Dialog Box (p. 2487). Specify the **Adjacent Dynamic Face Zone** and the **Direction Parameter**. This can only work on zones that are suited for layering (see Applicability of the Dynamic Layering Method (p. 664)).

10.6.8.2.16. Removing a Cell Layer

To manually remove a cell layer from an existing cell zone, select **Remove Cell Layer** in the **Type** drop-down list in the Define Event Dialog Box (p. 2487). Specify the **Adjacent Dynamic Face Zone** and the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Direction Parameter. This can only work on zones that are suited for layering (see Applicability of the Dynamic Layering Method (p. 664)).

10.6.8.2.17. Executing a Command

To execute a command, select **Execute Command** in the **Type** drop-down list in the Define Event Dialog Box (p. 2487) (Figure 10.76: The Define Event Dialog Box (p. 715)). A command can be a series of text or Scheme commands, or a macro you have defined (or will define) using the Define Macro Dialog Box (p. 2536) (see Defining Macros (p. 1640)). Enter the series of commands or the name of the macro in the **Command** text-entry box.

Important

If the command to be executed involves saving a file, see Saving Files During the Calculation (p. 1642) for important information.

10.6.8.2.18. Replacing the Mesh

To replace the mesh and interpolate existing data onto the new mesh during your simulation, select **Replace Mesh** from the **Type** drop-down list in the **Define Event** dialog box (Figure 10.76: The Define Event Dialog Box (p. 715)). Then, specify the replacement mesh under **Mesh File**. Enable **Interpolate Data Across Zones** if necessary (see Replacing the Mesh (p. 222) for details).

10.6.8.2.19. Resetting Inert EGR

To convert burnt gases at the end of the cycle to inert for the next cycle, select **Inert EGR Reset** from the **Type** drop-down list in the **Define Event** dialog box (Figure 10.76: The Define Event Dialog Box (p. 715)). Specify the **Zone(s)**. For further details, see Resetting Inert EGR (p. 1077).

10.6.8.2.20. Diesel Unsteady Flamelet Reset

To simulate multiple-cycle internal combustion engines using diesel unsteady flamelets, select **Diesel Unsteady Flamelet Reset** from the **Type** drop-down list in the **Define Event** dialog box (Figure 10.76: The Define Event Dialog Box (p. 715)). Specify the **Zone(s)**. This event is only applicable to and available with diesel unsteady flamelets with two or more flamelets. For further details, see Resetting Diesel Unsteady Flamelets (p. 1041). Note that **Diesel Unsteady Flamelet Reset** is generally preferable to **Inert EGR Reset** since an additional transport equation is avoided.

10.6.8.3. Exporting and Importing Events

If you want to save the events you have defined to a file, click **Write...** in the Dynamic Mesh Events Dialog Box (p. 2486) and specify the **Event File** in The Select File Dialog Box (p. 19).

To read the events back into ANSYS Fluent, click **Read...** in the Dynamic Mesh Events Dialog Box (p. 2486) and specify the **Event File** in The Select File Dialog Box (p. 19).

10.6.9. Specifying the Motion of Dynamic Zones

You must define the motion of the dynamic zones in your model. If the zone is a rigid body, you can use a profile or user-defined function (UDF) to define the motion of the rigid body or use the six DOF solver. If the zone is a deforming zone, you can define the geometry and the parameters that control the face or zone remeshing, if applicable. For a zone that is deforming and moving at the same time,

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

you can use a user-defined function to define the geometry and motion of the zone as they change with time.

10.6.9.1. General Procedure

You will specify the motion of the dynamic zones in your model using the **Dynamic Mesh Zones** dialog box

Setup → **Dynamic Mesh** → **Create/Edit...**

Details about specifying different types of motion are provided in this section.

10.6.9.1.1. Creating a Dynamic Zone

When you have completed the specification of a dynamic zone, click **Create** in the Dynamic Mesh Zones Dialog Box (p. 2490) to complete the specification and add the zone to the **Dynamic Mesh Zones** list.

10.6.9.1.2. Modifying a Dynamic Zone

If you want to make a change to the specification of a dynamic zone, select the zone in the **Dynamic Mesh Zones** list, change the specification, and then click **Create** in the Dynamic Mesh Zones Dialog Box (p. 2490) to update the specification.

10.6.9.1.3. Checking the Center of Gravity

If a dynamic zone has solid body motion, you can view its current position and orientation of the center of gravity (with respect to initial data) by selecting the zone in the **Dynamic Mesh Zones** list and viewing the values under **Center of Gravity Location** and **Rigid Body Orientation**.

10.6.9.1.4. Deleting a Dynamic Zone

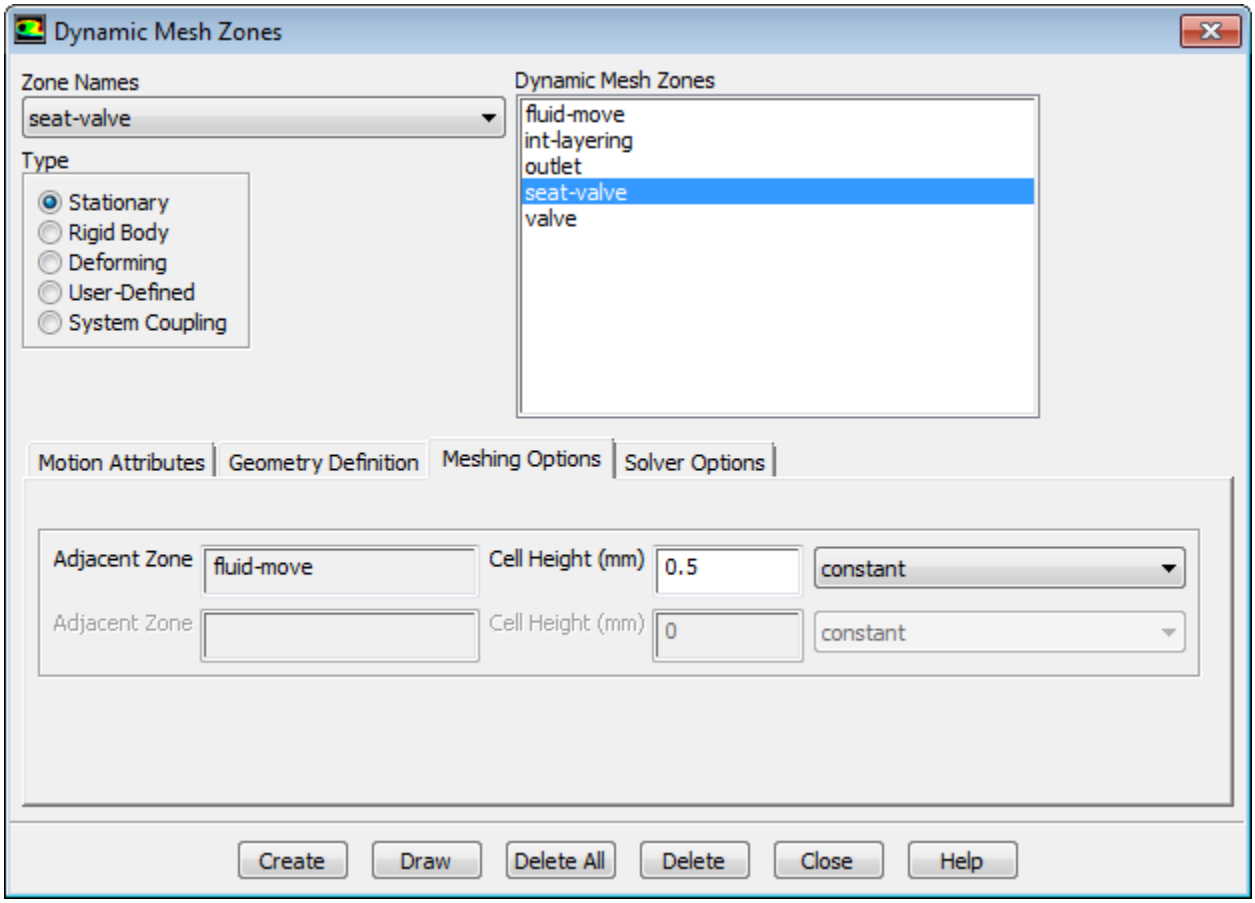
To delete a dynamic zone that you have specified, select the zone in the **Dynamic Mesh Zones** list, and click **Delete** or **Delete All**. The zone or zones will be removed from the **Dynamic Mesh Zones** list.

10.6.9.2. Stationary Zones

By default, if no motion (moving or deforming) attributes are assigned to a face or cell zone, then the zone is not considered when updating the mesh to the next time step. However, there are cases where an explicit declaration of a stationary zone is required. For example, if a cell zone is assigned some solid body motion, the positions of all nodes belonging to the cell zone will be updated even though some of the nodes may also be part of a non-moving boundary zone. An explicit declaration of a stationary zone excludes the nodes on these zones when updating the node positions.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.83: The Dynamic Mesh Zones Dialog Box for a Stationary Zone



To define a stationary zone in your model, follow the steps below.

1. Select the stationary zone in the **Zone Names** drop-down list.

2. Select **Stationary** under **Type**.

3. If the stationary zone is a face zone that is not a boundary of a CutCell dynamic cell zone, then define the **Cell Height** in the **Meshing Options** tab for any **Adjacent Zone** that is involved in local remeshing or dynamic layering. The **Cell Height** specifies the ideal height (h_{ideal} in Equation 10.15 (p. 662) and Equation 10.16 (p. 663) of the adjacent cells. Make a selection in the **Cell Height** drop-down menu to specify this value as either a **constant** or a compiled user-defined function.

   If you select the **constant** option, enter a value in the **Cell Height** text-entry box.

   If you choose to use a compiled user-defined function to define an ideal cell height that varies as a function of time or crank angle, you must first define a `DEFINE_DYNAMIC_ZONE_PROPERTY` UDF. After you have compiled the UDF source file, built a shared library, and loaded it into ANSYS Fluent, the name of the UDF library will be available for selection in the **Cell Height** drop-down list.

   Refer to the Fluent Customization Manual for information about UDFs.

4. If the stationary zone is a boundary of a CutCell dynamic cell zone (this is automatically detected if the CutCell dynamic cell zone is created first), then enter values for the **Maximum Mesh Size** and **Growth Rate** in the **Meshing Options** tab. If you have enabled **Inflation Layers** for the cell zone, then you will also have the option to enable **Zonal Inflation Layer Control** on the boundary zone and specify local in-

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

flation settings in the **Inflation Settings** dialog box (using the **Settings...** button). See Figure 10.84: The Dynamic Mesh Zones Dialog Box for a CutCell Boundary Zone (p. 726) for an example of the **Dynamic Mesh Zones** dialog box with the **Meshing Options** tab as it appears for CutCell boundary zones. (The same meshing input is required at all CutCell boundary zones, whether the motion type is **Stationary**, **Rigid Body**, **Deforming**, or **User-Defined**.)

Two different size functions are available to control the local mesh refinement:

- soft size function

  For this size function, the Cartesian mesh is locally refined such that the resulting mesh size is smaller or equal to the **Maximum Mesh Size** specified.

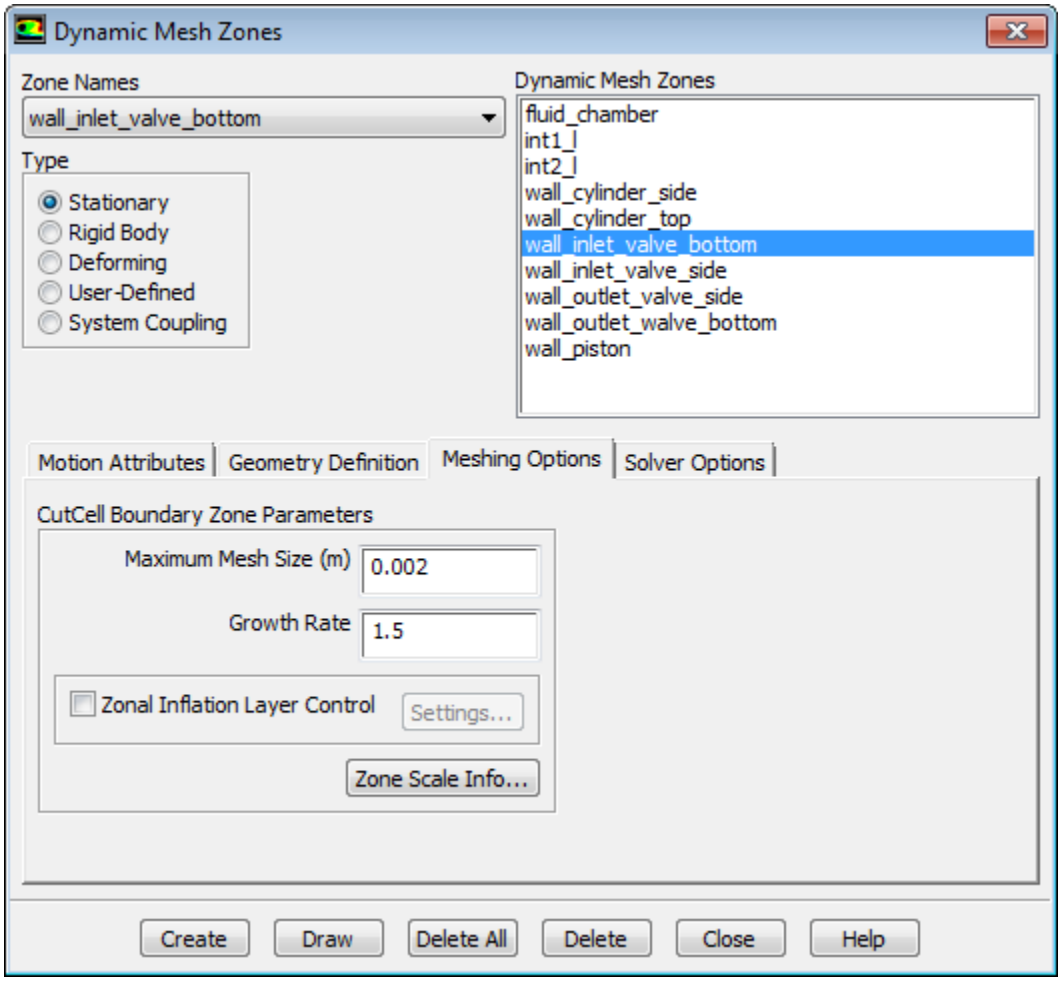
- mesh-based size function

  For this size function, the Cartesian mesh is locally refined such that the resulting mesh size is locally of the same size as the input mesh provided.

The switch between using the soft or the mesh-based size function depends on the value entered for the **Maximum Mesh Size**. A value of `0` specifies that the mesh-based size function is used. If a positive value is entered, then the soft size-function is used and the value entered locally limits the mesh size. The **Growth Rate** controls the rate at which the Cartesian mesh grows away from the boundary. The maximum Cartesian mesh size obtained in the cell zone is always controlled by the specified **Maximum Mesh Size** for the CutCell cell zone.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.84: The Dynamic Mesh Zones Dialog Box for a CutCell Boundary Zone



You can view the vital statistics of your zone by clicking on the **Zone Scale Info...** button. This opens the Zone Scale Info Dialog Box (p. 2499), where you can view the minimum, maximum, and average length scale values, as well as the maximum skewness value.

5. Click **Create**.

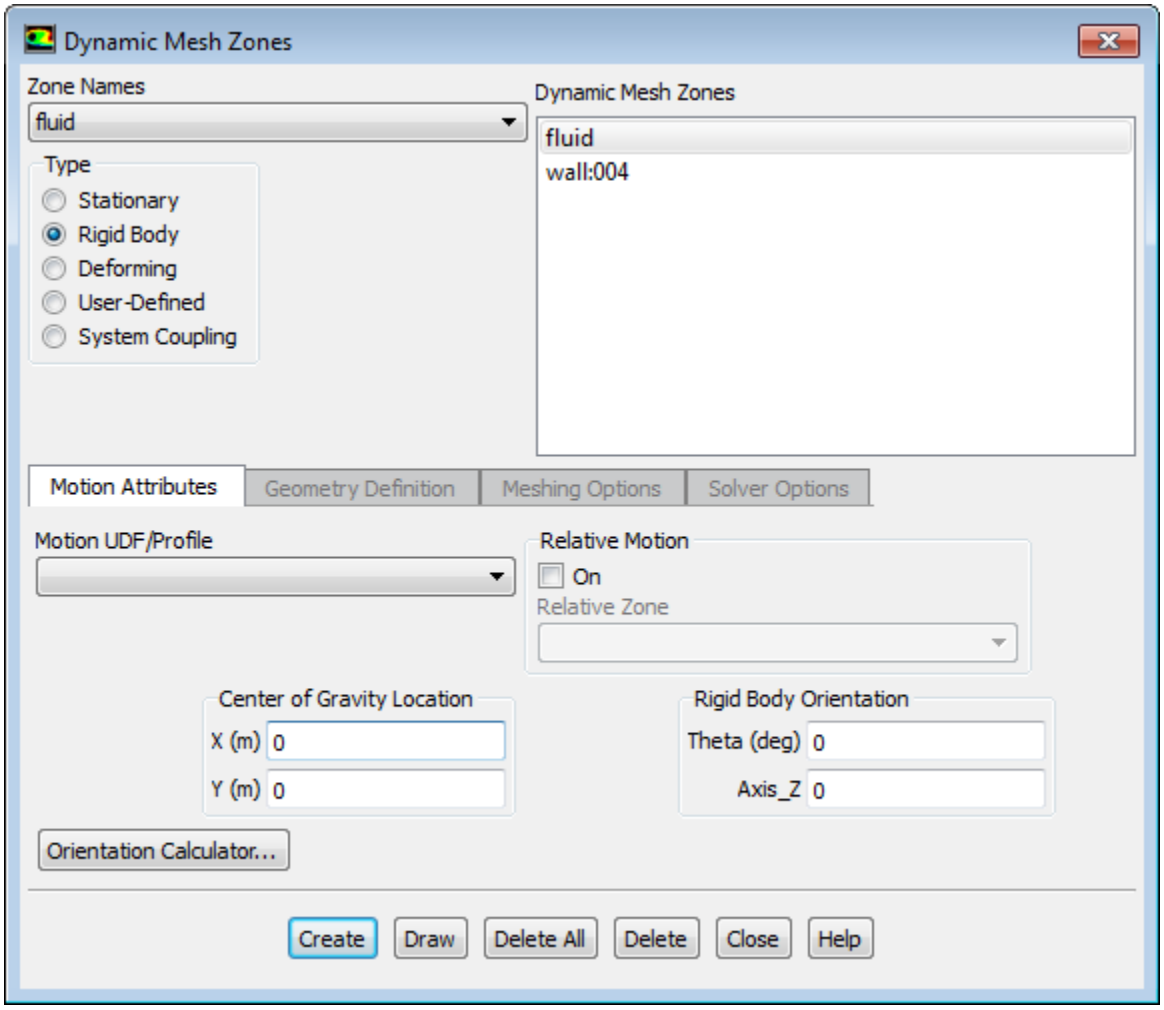
   Note that if you specify inflation layer parameters in the **Inflation Settings** dialog box, these settings are not saved until you create or update the dynamic zone.

10.6.9.3. Rigid Body Motion

To define a rigid-body zone in your model, follow the steps below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.85: The Dynamic Mesh Zones Dialog Box for a Rigid Body Motion



1. Select the rigid body zone in the **Zone Names** drop-down list.

2. Select the **Rigid Body** option under **Type**.

3. If you want to specify the motion of the rigid body zone using a profile or user-defined function, then select a profile or user-defined function from the **Motion UDF/Profile** drop-down list in the **Motion Attributes** tab. See Profiles (p. 435) and Solid-Body Kinematics (p. 739) for information on profiles, and see the Fluent Customization Manual for information on user-defined functions.

4. If you enabled the **In-Cylinder** option in the Dynamic Mesh Task Page (p. 2471), ANSYS Fluent provides built-in functions in the **Motion UDF/Profile** drop-down list that can be useful for defining the rigid body motion of a piston. If you would like the motion of the piston to be a function of crank angle (that is, governed by Equation 10.25 (p. 692)), then you should select ****piston-full****. For further information, see In-Cylinder Settings (p. 690).

5. If you enabled the **Six DOF** option in the Dynamic Mesh Task Page (p. 2471), make sure that **On** is enabled in the **Six DOF** group box in the **Motion Attributes** tab (see Figure 10.87: The Dynamic Mesh Zones Dialog Box for a Rigid Body Motion Using the Six DOF Solver (p. 730)), to ensure that the six DOF solver is being used. Then make a selection from the **Six DOF UDF/Properties** drop-down list; for details on creating the selections available in this list, Setting Rigid Body Motion Attributes for the Six DOF Solver (p. 707).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that the **Passive** option in the **Six DOF** group box is used when you do not want the forces and moments on the zone to be taken into consideration.

6. Specify the initial location of the center of gravity for the rigid body by entering the coordinates of the center of gravity in **Center of Gravity Location**. Note that for one DOF dynamic zones, this field will be overwritten when you click **Create** by the **Center of Rotation** values you specified in the Six DOF Properties Dialog Box (p. 2482).

7. If you want a rigid body to move relative to another rigid body, then you must enable **Relative Motion** and specify the **Relative Zone**.

   Note

   You must create the relative rigid body zone before creating a rigid body that is dependent on that zone, if you are setting up the motion using the graphical user interface (GUI). The order in which you specify the zones does not matter if you set up the motion using the text user interface (TUI).

8. Specify the orientation of the rigid body (in the inertial coordinate system) by entering the orientations of the rigid body in **Rigid Body Orientation**.

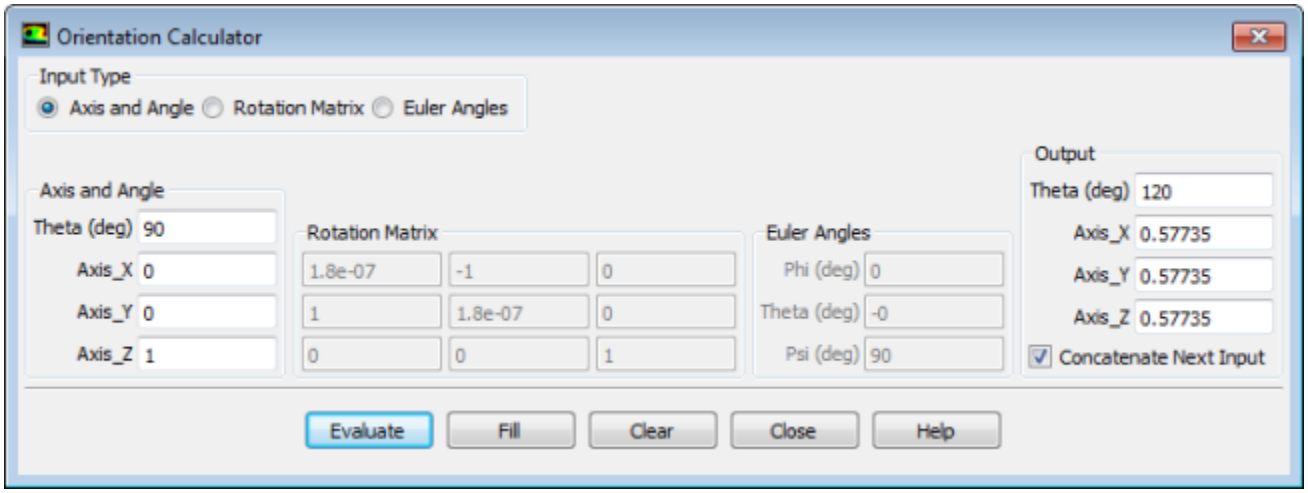
   Together, the **Rigid Body Orientation** *and the* **Center of Gravity Location** *are the local reference frame that you can choose to use for your user-defined function (UDF) or set of six DOF properties that describes the motion of the rigid body. For additional information about creating UDFs for solid-body kinematics, see* `DEFINE_CG_MOTION` *in the Fluent Customization Manual; for details on creating UDFs / properties for the six DOF solver, see Setting Rigid Body Motion Attributes for the Six DOF Solver (p. 707).*

   For most cases, this is an initial reference orientation that ANSYS Fluent later updates, letting you keep track of the object's current orientation. The rigid body orientation is most useful when using the six DOF solver, where it is used to compute the transformation matrices (Six DOF Solver Theory in the Theory Guide).

   If needed, you can use the **Orientation Calculator** to convert a rotation matrix or Euler angles into the required axis and angle inputs. You can also use this calculator if you know the rotations the rigid body will experience, but do not know its final orientation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.86: Orientation Calculator Dialog Box



To use the **Orientation Calculator**:

a. Enter a value, either in **Axis and Angle**, **Rotation Matrix**, or **Euler Angle** format, and click **Evaluate** to see the value in the output format.

The calculator also converts the entered values into the other two "grayed-out" input formats.

> **Note**
>
> There are several possible Euler angle representations. Fluent uses a vector defined by ϕ, θ, and ψ where: ψ is the rotation about the original Z axis, θ is the rotation about the rotated Y axis, and ϕ is the rotation about the rotated X axis.

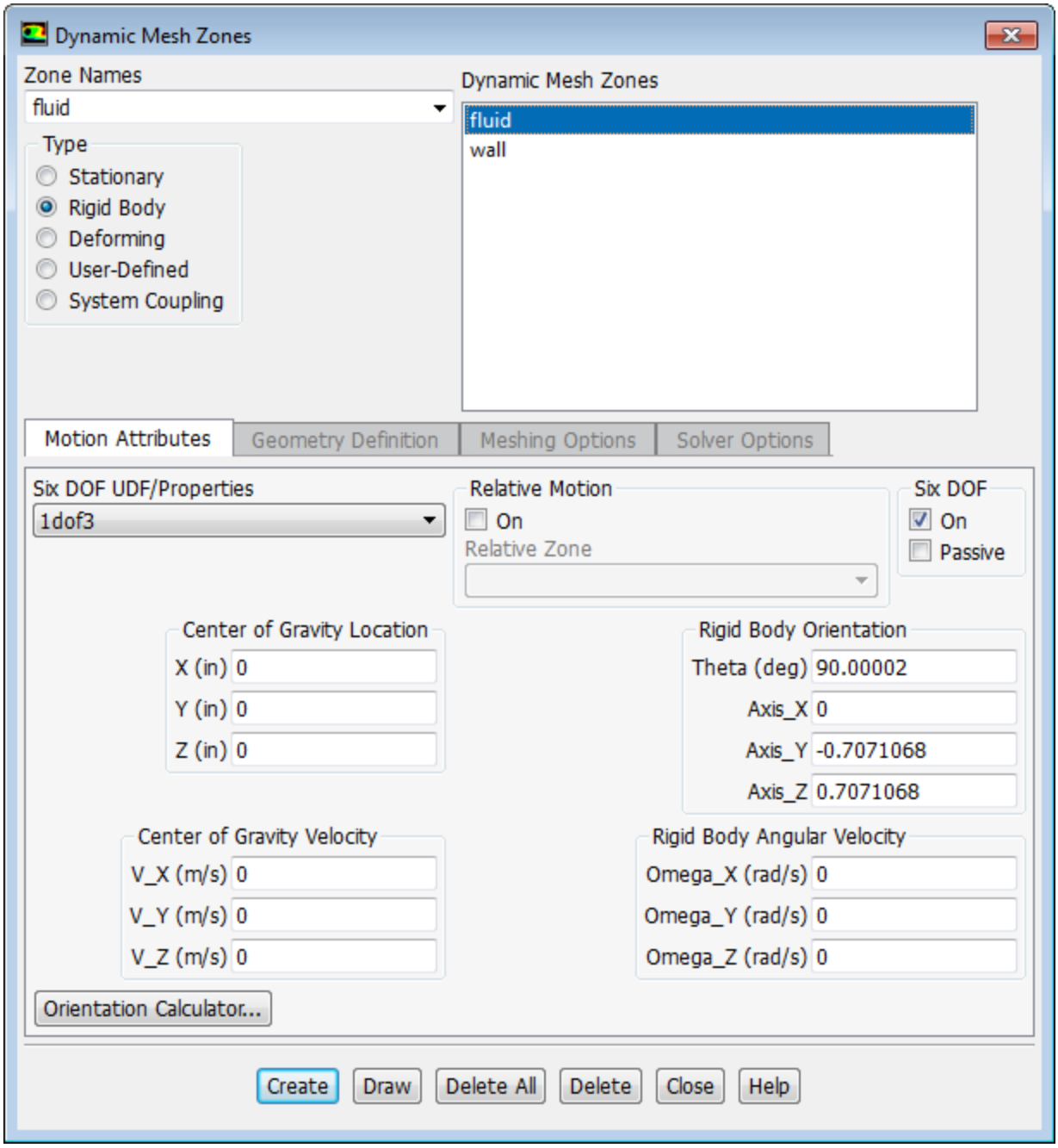
b. Enable **Concatenate Next Input** if you want the next input to be concatenated to the previous output.

c. Click **Fill** to use the current **Output** as your **Rigid Body Orientation** in the **Dynamic Mesh Zones** dialog box.

9. When using the six DOF solver, specify the velocity of the center of gravity with respect to the inertia coordinate system by entering the velocity of the center of gravity in **Center of Gravity Velocity**. Also, specify the angular velocity of the rigid body with respect to the inertia coordinate system by entering the angular velocity of the rigid body in **Rigid Body Angular Velocity**. Note that these fields are ignored for one DOF rotation and translation, respectively.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.87: The Dynamic Mesh Zones Dialog Box for a Rigid Body Motion Using the Six DOF Solver



10. If you are solving an in-cylinder problem, specify the direction of the reference axis of the valves or piston in **Valve/Piston Axis**.

    The current valve lift or piston stroke is automatically updated in **Lift/Stroke** when you click **Create** based on the parameters you have specified earlier when you first invoke the in-cylinder option.

11. By default, the boundary mesh motion is taken into consideration when imposing the physical boundary conditions, even if the boundary moves because of a moving adjacent cell zone and no dynamic zone has been created for the boundary. If this is not the desired behavior, and if you would like to exclude the mesh motion from contributing to the boundary conditions, then you need to enable the **Exclude Mesh Motion in Boundary Conditions** option for that zone.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12. If the rigid body zone is a face zone that is not a boundary of a CutCell dynamic cell zone, specify the **Cell Height** for each **Adjacent Zone** in the **Meshing Options** tab. The **Cell Height** is the ideal cell height (h_{ideal} in Equation 10.15 (p. 662) and Equation 10.16 (p. 663)) that is used by ANSYS Fluent to determine when the prismatic layer next to the rigid body should be split or merged with the layer next to it. If the adjacent zone is tetrahedral or triangular, the ideal height is used by ANSYS Fluent to determine if adjacent cells must be agglomerated for local remeshing. Make a selection in the **Cell Height** drop-down menu to specify this value as either a **constant** or a compiled user-defined function.

    If you select the **constant** option, enter a value in the **Cell Height** text-entry box.

    If you choose to use a compiled user-defined function to define an ideal cell height that varies as a function of time or crank angle, you must first define a `DEFINE_DYNAMIC_ZONE_PROPERTY` UDF. After you have compiled the UDF source file, built a shared library, and loaded it into ANSYS Fluent, the name of the UDF library will be available for selection in the **Cell Height** drop-down list.

    Refer to the Fluent Customization Manual for information about UDFs.

13. If the rigid body zone is a boundary of a CutCell dynamic cell zone (this is automatically detected if the CutCell dynamic cell zone is created first), then enter values for the **Maximum Mesh Size** and **Growth Rate** in the **Meshing Options** tab, as described in step 4 of Stationary Zones (p. 723). If you have enabled **Inflation Layers** for the cell zone, then you will also have the option to enable **Zonal Inflation Layer Control** on the boundary zone and specify local inflation settings in the **Inflation Settings** dialog box (using the **Settings...** button).

14. If the dynamic zone is a face zone with an adjacent boundary layer mesh, you must apply boundary layer smoothing using the **Deform Adjacent Boundary Layer with Zone** option in the **Meshing Options** tab if you want the boundary layer to move with the moving face zone. For example, this option is necessary if you are applying face region remeshing with prism layers and you have not decomposed the mesh volume (see Face Region Remeshing with Wedge Cells in Prism Layers (p. 678)). In such circumstances, the **Deform Adjacent Boundary Layer with Zone** option ensures that the base prism elements shown in Figure 10.43: Volume Decomposition for the Base of the Prism Layers (p. 680) will move rigidly with the piston.

    Note that the boundary layer smoothing method is primarily intended for translational motion of the face zone. If significant rotation is applied to the face zone, the boundary layer cells may become skewed. For more information refer to Boundary Layer Smoothing Method (p. 658).

15. In regions using the six DOF solver, solution stabilization may be necessary if the use of implicit mesh update is insufficient to stabilize the solver due to the fluid motion. Solution stabilization options for the six DOF region can be set by opening the **Solver Options** tab, selecting the **Solution Stabilization** check box, and then specifying the appropriate options under **Stabilization Parameters**. Available options include the **Scale Factor** and the **Method**, which can be either **volume-based** (default) or **coefficient-based**. See Solution Stabilization for System Coupling and Six DOF Motion (p. 738) for details. The **Scale Factor** must be set to a nonzero value for the stabilization to have an effect.

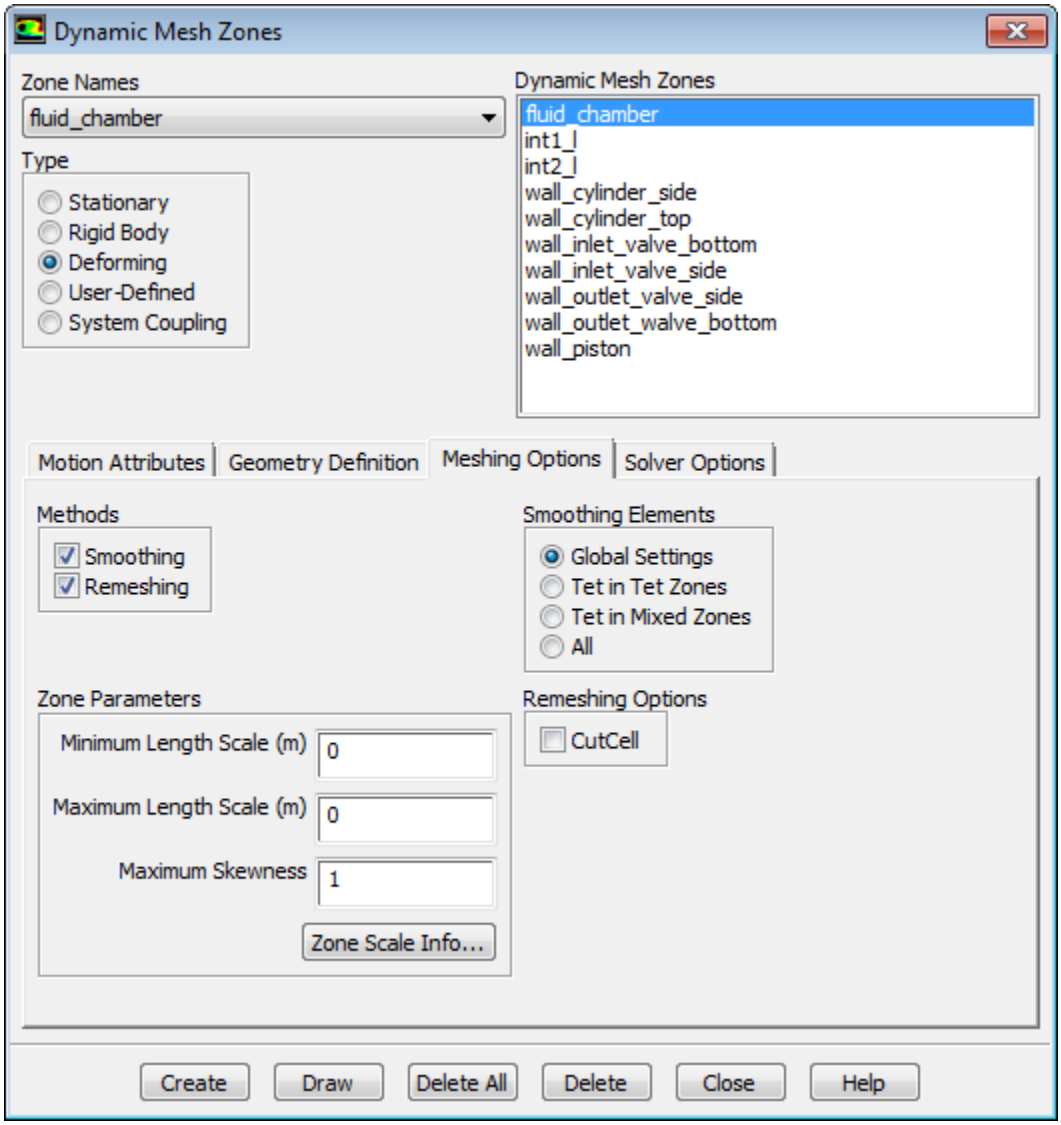
    These options can also be set using the text user interface commands for the `rigid body` region type when the six DOF solver is enabled, and solver option settings for each dynamic zone are visible when you use the `list` command.

16. Click **Create**.

10.6.9.4. Deforming Motion

To define a deforming zone in your model, follow the steps below.

Figure 10.88: The Dynamic Mesh Zones Dialog Box for a Deforming Motion with Cell Zone Options



1. Select the deforming zone in the **Zone Names** drop-down list.
2. Select the **Deforming** option under **Type**.
3. Specify the geometry of the deforming zone in the **Geometry Definition** tab.
   - If no geometry is available, select **faceted** in the **Definition** drop-down list.
   - If the geometry is a plane, select **plane** in the **Definition** drop-down list. To define the plane, enter the position of a point on the plane in **Point on Plane** and the plane normal in **Plane Normal**.
   - If the geometry is a cylinder, select **cylinder** in the **Definition** drop-down list. To define the cylinder, enter the **Cylinder Radius**, the **Cylinder Origin** and the **Cylinder Axis**.
   - If the geometry is unspecified and mesh motion normal to the boundary is permissible, select **unspecified** in the **Definition** drop-down list. This option is only available if the linearly elastic solid mesh smoothing method is enabled (Linearly Elastic Solid Based Smoothing Method (p. 656)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If the geometry is described by a user-defined function, select **user-defined** in the **Definition** drop-down list and the appropriate user-defined functions in the **Geometry UDF** drop-down list. See the Fluent Customization Manual for information on user-defined functions.

For 3D simulations, ANSYS Fluent allows you to preserve features on the boundary zones that are important for the flow. Such features may be at the juncture of different boundary zones, or they may be on a single non-planar boundary zone. For any geometry definition (**faceted**, **plane**, **cylinder**, or **user-defined**), you can indicate whether you want to preserve features of a specific angular range by selecting **Include Features** under **Feature Detection** and setting a threshold value for the **Feature Angle** in degrees. For more information (including how the **Feature Angle** is defined), see Feature Detection (p. 690).

When available, the geometry information is used to project nodes on the deforming zone after remeshing the face zone, or if nodes are moved through smoothing.

4. For deforming face and cell zones that are not CutCell zones, specify the appropriate remeshing parameters in the **Meshing Options** tab.

   You can locally disable or enable **Smoothing** and/or **Remeshing** and use any **Smoothing Methods** and/or **Remeshing Methods**.

   You can view the vital statistics of your zone by clicking the **Zone Scale Info...** button. This opens the Zone Scale Info Dialog Box (p. 2499), where you can view the minimum and maximum length scale values, as well as the maximum skewness values.

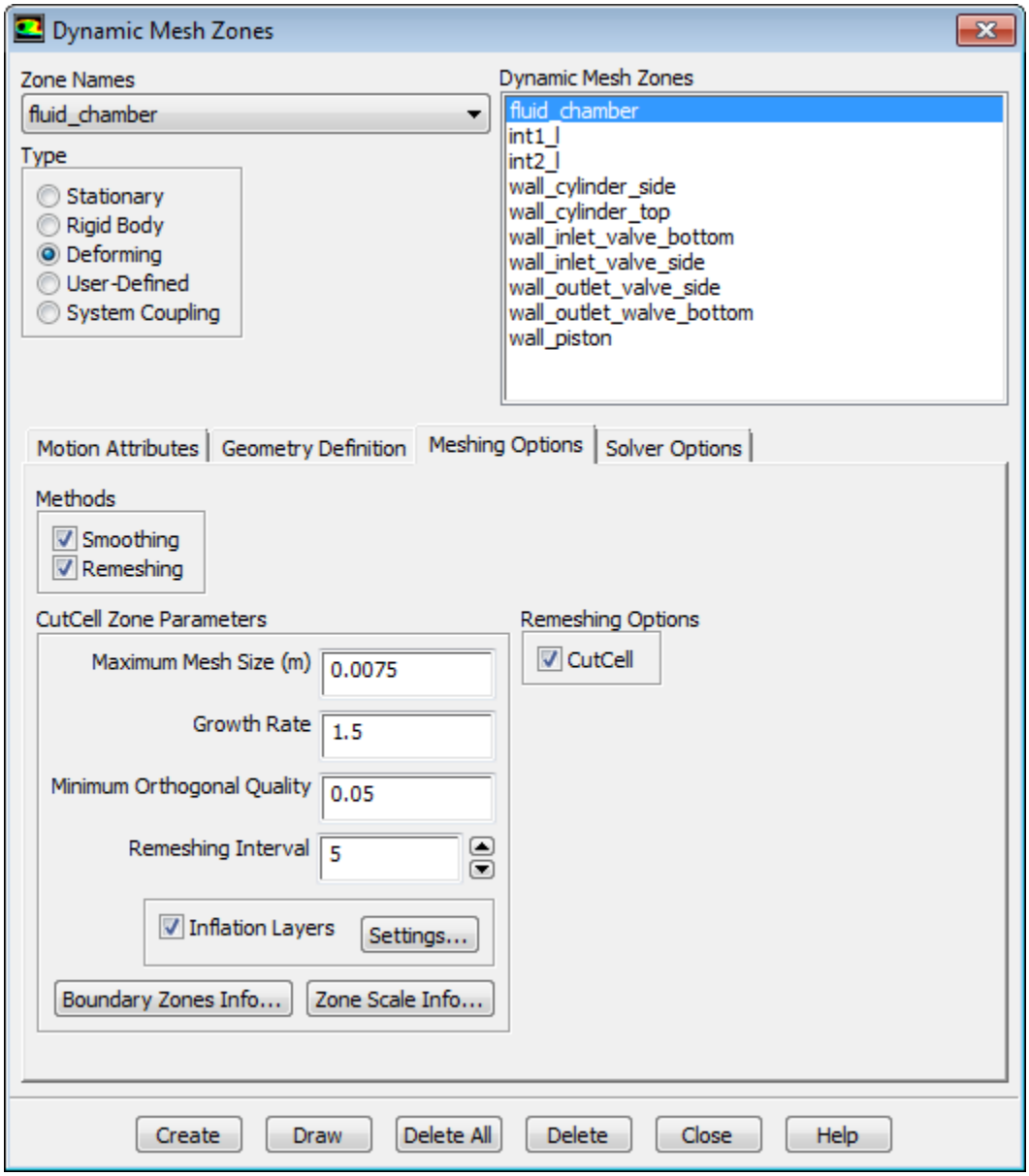
   You can revise the **Minimum Length Scale**, **Maximum Length Scale** and/or **Maximum Skewness** if you want to specify local values for this dynamic zone that differ from those you specified globally in the Mesh Method Settings Dialog Box (p. 2474). Note that the global settings may still be used if ANSYS Fluent determines that your local settings are unreasonable. You should use the information found in the Zone Scale Info Dialog Box (p. 2499) in order to set your values.

   If spring-based smoothing is used and you selected a cell zone, then you have the option to overwrite the element type selection made in the **Mesh Method Settings** dialog box (see Figure 10.14: The Smoothing Tab of the Mesh Method Settings Dialog Box (3D) (p. 645)). Select **Global Settings** if no zone specific smoothing element type is desired.

5. To specify that a deforming cell zone is remeshed using the CutCell zone method, enable the **CutCell** option in the **Remeshing Options** group box of the **Meshing Options** tab (see Figure 10.89: The Dynamic Mesh Zones Dialog Box for a Deforming CutCell Cell Zone (p. 734)). Note that this option is only available if you have previously enabled the **CutCell Zone** option in the **Remeshing** tab of the **Mesh Method Settings** dialog box (see Figure 10.33: The Remeshing Tab in the Mesh Method Settings Dialog Box (p. 667)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.89: The Dynamic Mesh Zones Dialog Box for a Deforming CutCell Cell Zone



After the **CutCell** option has been enabled, you must enter values for the global CutCell parameters **Maximum Mesh Size** and **Growth Rate**, as well as the controls for the remeshing frequency, **Minimum Orthogonal Quality** and **Remeshing Interval**. You also have the option to enable **Inflation Layers** for the cell zone and specify the global inflation parameters in the **Inflation Settings** dialog box. See Using the CutCell Zone Remeshing Method (p. 683) for more information about adding inflation layers to a CutCell mesh.

The **Maximum Mesh Size** specified for the CutCell cell zone controls the global maximum size of the Cartesian cells. The global **Growth Rate** is used for all mesh-based size functions applied to CutCell boundary zones that are not defined as dynamic zones. When selecting values for these controls, it may be helpful to view the statistics of your cell zone by clicking on the **Zone Scale Info...** button. This opens the Zone Scale Info Dialog Box (p. 2499), where you can view the minimum and maximum length scale values, as well as the maximum skewness values.

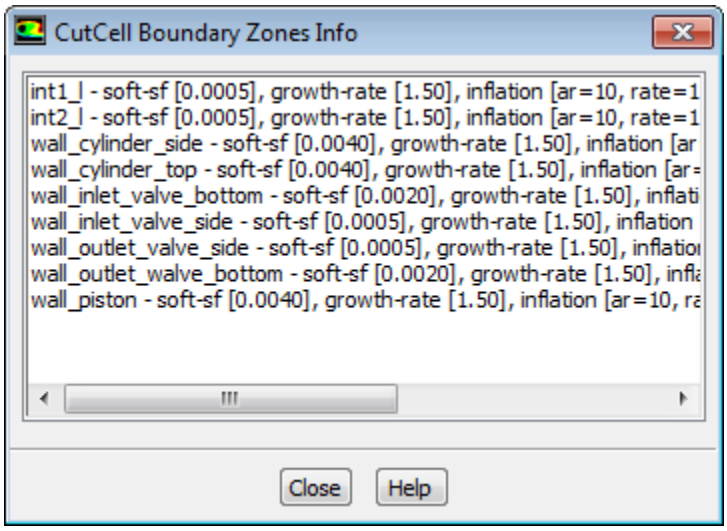
The CutCell remeshing is performed automatically whenever cells exhibit an orthogonal quality (as defined in Mesh Quality (p. 131)) that is less than the specified **Minimum Orthogonal Quality**. The

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

remeshing is also performed periodically after the calculation has undergone the number of time steps you specified for the **Remeshing Interval**.

Click the **Boundary Zones Info...** button to open the **CutCell Boundary Zones Info** dialog box (Figure 10.90: The CutCell Boundary Zones Info Dialog Box (p. 735)). This dialog box lists all the boundary zones associated with the CutCell cell zone, along with the associated size functions and zonal meshing parameters. The inflation layer settings used on all boundaries are also shown. It is recommended that you revisit this dialog box after you have created all of the dynamic zones, in order to review all of the parameters you have set.

Figure 10.90: The CutCell Boundary Zones Info Dialog Box



6. If the deforming zone is a boundary of a CutCell dynamic cell zone (this is automatically detected if the CutCell dynamic cell zone is created first), then enter values for the **Maximum Mesh Size** and **Growth Rate** in the **Meshing Options** tab, as described in step 4 of Stationary Zones (p. 723). If you have enabled **Inflation Layers** for the cell zone, then you will also have the option to enable **Zonal Inflation Layer Control** on the boundary zone and specify local inflation settings in the **Inflation Settings** dialog box (using the **Settings...** button).

7. Click **Create**.

10.6.9.5. User-Defined Motion

For a zone that is deforming and/or moving, you can define the position of each node on the general deforming / moving zone using a user-defined function (UDF). To define a moving / deforming zone, follow the steps below.

1. Select the moving and deforming zone in the **Zone Names** drop-down list.

2. Select the **User-Defined** option under **Type**.

3. In the **Motion Attributes** tab, select the user-defined function that defines the geometry and motion of the zone from the **Mesh Motion UDF** drop-down list. See the Fluent Customization Manual for information on user-defined functions used to specify user-defined motion.

4. By default, the boundary mesh motion is taken into consideration when imposing the physical boundary conditions, even if the boundary moves because of a moving adjacent cell zone and no dynamic zone has

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

been created for the boundary. If this is not the desired behavior, and if you would like to exclude the mesh motion from contributing to the boundary conditions, then you need to enable the **Exclude Mesh Motion in Boundary Conditions** option for that zone.

5. If you want the movement / deformation to be relative to the rigid body motion of a cell zone, then you can enable **Relative Motion** and select the cell zone from the **Relative Zone** list.

   Note

   You must specify the movement / deformation in your UDF using `NODE_COORD_NEST`, rather than specify the final coordinates (as you would if you were NOT specifying it relative to a cell zone). For additional information, see `DEFINE_GRID_MOTION` in the *Fluent Customization Manual*.

   You must create the relative rigid body zone before creating the user-defined zone if you are setting up the zones using the graphical user interface (GUI). The order in which you create the zones does not matter if you set up the zones using the text user interface (TUI).

6. For a face zone that is not a boundary of a CutCell dynamic cell zone, you can specify the **Cell Height** in the **Meshing Options** tab for any **Adjacent Zone** that is involved in local remeshing or dynamic layering. The **Cell Height** specifies the ideal height (h_{ideal} in Equation 10.15 (p. 662) and Equation 10.16 (p. 663) of the adjacent cells. Make a selection in the **Cell Height** drop-down menu to specify this value as either a **constant** or a compiled user-defined function.

   If you select the **constant** option, enter a value in the **Cell Height** text-entry box.

   If you choose to use a compiled user-defined function to define an ideal cell height that varies as a function of time or crank angle, you must first define a `DEFINE_DYNAMIC_ZONE_PROPERTY` UDF. After you have compiled the UDF source file, built a shared library, and loaded it into ANSYS Fluent, the name of the UDF library will be available for selection in the **Cell Height** drop-down list.

   Refer to the separate Fluent Customization Manual for information about UDFs.

7. If the face zone is a boundary of a CutCell dynamic cell zone (this is automatically detected if the CutCell dynamic cell zone is created first), then enter values for the **Maximum Mesh Size** and **Growth Rate** in the **Meshing Options** tab, as described in step 4 of Stationary Zones (p. 723). If you have enabled **Inflation Layers** for the cell zone, then you will also have the option to enable **Zonal Inflation Layer Control** on the boundary zone and specify local inflation settings in the **Inflation Settings** dialog box (using the **Settings...** button).

8. If the dynamic zone is a face zone with an adjacent boundary layer mesh, and you want to use the boundary layer smoothing method (as described in Boundary Layer Smoothing Method (p. 658)), enable the **Deform Adjacent Boundary Layer with Zone** option in the **Meshing Options** tab.

9. For a 3D boundary zone, you can allow local face remeshing by enabling the **Local** option (under **Remeshing Methods)** in the **Meshing Options** tab. For details, see Local Face Remeshing Method (p. 669). Enabling this option allows you to do the following:

   - You can revise the **Maximum Skewness** allowed for faces if you want to specify a local value for this dynamic zone that differs from the one you specified globally in the Mesh Method Settings Dialog

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Box (p. 2474). Note that the global setting may still be used if ANSYS Fluent determines that your local setting is unreasonable.

- You can preserve features, not only at the juncture of different boundary zones, but also on a single non-planar boundary zone. In the **Geometry Definition** tab, you can indicate whether you want to include features of a specific angular range by selecting **Include Features** under **Feature Detection** and setting a threshold value for the **Feature Angle** in degrees. For more information (including how the **Feature Angle** is defined), see Feature Detection (p. 690).

10. Click **Create**.

10.6.9.5.1. Specifying Boundary Layer Deformation Smoothing

For a boundary layer that deforms according to the adjacent face zone, the zone that is deforming and moving is usually defined using a user-defined function (UDF), as described in User-Defined Motion (p. 735). To define a moving and deforming boundary layer, perform the steps that follow.

If the boundary layer borders a face zone that is only moving and is not deforming, you should consider applying rigid body motion to the face zone (see Rigid Body Motion (p. 726)) rather than user-defined motion, as rigid body motion usually involves a simpler UDF.

1. Set up the moving and deforming zone.

   a. Select the moving and deforming zone in the **Zone Names** drop-down list.

   b. Select the **User-Defined** option under **Type**.

   c. In the **Motion Attributes** tab, select the user-defined function that defines the geometry and motion of the zone from the **Mesh Motion UDF** drop-down list.

   d. In the **Meshing Options** tab, enable the **Deform Adjacent Boundary Layer with Zone** option.

   e. Click **Create**.

2. Set up a deforming dynamic zone for the fluid zone that contains the boundary layer.

   a. Select the fluid zone that contains the boundary layer from the **Zone Names** drop-down list.

   b. Select **Deforming** from **Type** list.

   c. In the **Meshing Options** tab, enable **Smoothing** and **Remeshing** in the **Methods** group box.

   d. Click **Create**.

3. If the fluid zone set up in the step 2 consists entirely of the boundary layer elements, set up a deforming dynamic zone for the neighboring fluid zone. This step is necessary because the deforming boundary layer will deform the adjacent cells.

   a. Select the fluid zone that neighbors the boundary layer zone from the **Zone Names** drop-down list.

   b. Select **Deforming** from **Type** list.

   c. In the **Meshing Options** tab, enable **Smoothing** and **Remeshing** in the **Methods** group box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

d. Click **Create**.

10.6.9.6. System Coupling Motion

For a zone that is involved in a system coupling, the motion is defined by the application that ANSYS Fluent is coupled with on this zone. For more details about setting up a simulation with system coupling see the Fluent in Workbench User's Guide and the System Coupling Guide. To define a system coupling region, follow the steps below.

1. Select the moving and deforming zone in the **Zone Names** drop-down list.

2. Select the **System Coupling** option under **Type**.

3. The system coupling region cannot be a boundary of a CutCell dynamic cell zone, so you can specify the **Cell Height** in the **Meshing Options** tab for any **Adjacent Zone** that is involved in local remeshing or dynamic layering. The **Cell Height** specifies the ideal height (h_{ideal} in Equation 10.15 (p. 662) and Equation 10.16 (p. 663) of the adjacent cells. Make a selection in the **Cell Height** drop-down menu to specify this value as either a **constant** or a compiled user-defined function.

   If you select the **constant** option, enter a value in the **Cell Height** text-entry box.

   If you choose to use a compiled user-defined function to define an ideal cell height that varies as a function of time or crank angle, you must first define a `DEFINE_DYNAMIC_ZONE_PROPERTY` UDF. After you have compiled the UDF source file, built a shared library, and loaded it into ANSYS Fluent, the name of the UDF library will be available for selection in the **Cell Height** drop-down list.

   Refer to the separate Fluent Customization Manual for information about UDFs.

4. Solution stabilization may be necessary to achieve convergence for system coupling cases where there is a strong coupling. Solution stabilization options for the system coupling region can be set by opening the **Solver Options** tab, selecting the **Solution Stabilization** check box, and then specifying the appropriate options under **Stabilization Parameters**. Available options include the **Scale Factor** and the **Method**, which can be either **volume-based** (default) or **coefficient-based**. The **Scale Factor** must be set to a nonzero value for the stabilization to have an effect. See Solution Stabilization for System Coupling and Six DOF Motion (p. 738) for details.

   These options can also be set using the text user interface commands for the `system coupling` region type, and solver option settings for each dynamic zone are visible when you use the `list` command.

5. Click **Create**.

Note

If the **System Coupling** option is enabled, and ANSYS Fluent is not involved with a system coupling simulation, then this zone type behaves in the same way as a stationary zone.

10.6.9.7. Solution Stabilization for System Coupling and Six DOF Motion

For cases with strong fluid-structure interaction, stabilization is achieved through a boundary source coefficient introduced in the continuity equation, designed to improve the diagonal dominance of the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

matrix system in the cells adjacent to system coupling or six DOF boundaries. Two methods for this boundary source coefficient are available:

- volume-based

  This method uses the cell volume V to re-scale the diagonal entry of the linear matrix system corresponding to the discretized continuity equation (a_{ij}) as $a_{ij,s}$:

  $$a_{ij,s}=a_{ij}+KV \quad \text{if } i=j \forall i,j \in \{1,2,...,n\} \tag{10.32}$$

- coefficient-based

  This method directly re-scales the diagonal entry of the linear matrix system corresponding to the discretized continuity equation (a_{ij}) as $a_{ij,s}$:

  $$a_{ij,s}=a_{ij}+Ka_{ij} \quad \text{if } i=j \forall i,j \in \{1,2,...,n\} \tag{10.33}$$

where K is the scale factor and n is the number of cells. Note that the value you choose for K will only affect the rate of convergence, and will not affect the converged solution.

10.6.9.8. Solid-Body Kinematics

ANSYS Fluent uses solid-body kinematics if the motion is prescribed based on the position and orientation of the center of gravity of a moving object. This is applicable to both cell and face zones.

The motion of the solid-body can be specified either as a profile or as a user-defined function (UDF). A profile may be defined by the following profile fields:

- time (time)
- crank angle (angle) (in-cylinder flows only)
- position (x, y, z)
- linear velocity (v_x, v_y, v_z)
- angular velocity (ω_x, ω_y, ω_z)
- orientation (θ_x, θ_y, θ_z)

For in-cylinder simulations, the velocity profiles for valves can be expressed as a function of crank angle instead of time. In addition, transient boundary condition profiles can also be expressed as a function of crank angle instead of time. For more information about transient profiles, see Defining Transient Cell Zone and Boundary Conditions (p. 448).

Below are two examples of a profile format:

```
((movement_linear 3 point)
(time
0  1  2 )
(x
2  3  4 )
(v_y
0  -5  0 )
)

((movement_angular 3 point)
(time
0  1  2 )
```

```
(omega_x
2  3  4 )
)
```

For in-cylinder flows, crank angles can be included in transient tables as well as transient profiles, in a similar fashion to time. An example of a transient table using (crank) angle is as follows:

```
example 2 3 1
angle temperature
0  300
180 500
360 300
```

An example of a transient profile using (crank) angle is as follows:

```
((example transient 3 1)
(angle
0.000000e+00 1.800000e+02 3.600000e+02)
(temperature
3.000000e+02 5.000000e+02 3.000000e+02)
)
```

In addition to the motion description, you must also specify the starting location of the center of gravity and orientation of the solid body. In 2D cases (and 3D cases that do not use the six DOF solver), ANSYS Fluent automatically updates the center of gravity position and orientation at every time step such that

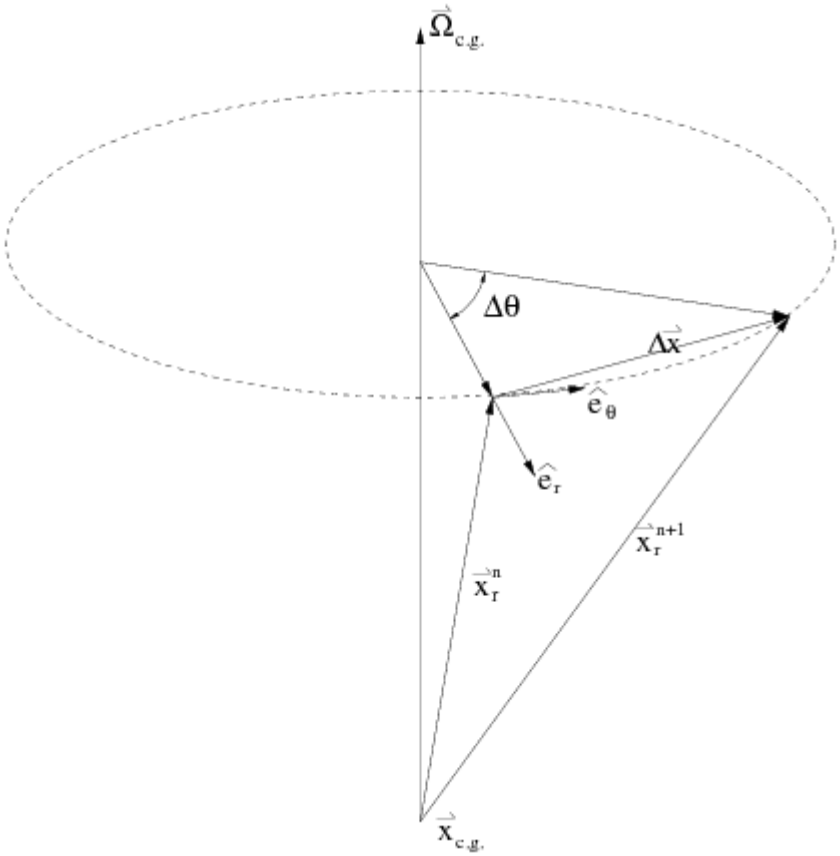
$$\begin{aligned}\vec{x}_{c.g.}^{\,n+1} &= \vec{x}_{c.g.}^{\,n} + \vec{v}_{c.g.}\Delta t \\ \vec{\theta}_{c.g.}^{\,n+1} &= \vec{\theta}_{c.g.}^{\,n} + \vec{\Omega}_{c.g.}\Delta t\end{aligned} \tag{10.34}$$

where $\vec{x}_{c.g.}$ and $\vec{\theta}_{c.g.}$ are the position and orientation of the center of gravity, $\vec{v}_{c.g.}$ and $\vec{\Omega}_{c.g.}$ are the linear and angular velocities of the center of gravity. 3D, six DOF cases use a more complex form of Equation 10.34 (p. 740) when updating θ.

Typically, $\vec{\theta}$ is chosen to be an appropriate set of Euler angles. In this case, the solid-body motion must be specified using a user-defined function (`DEFINE_CG_MOTION`).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.91: Solid Body Rotation Coordinates



The position vectors on the solid body are updated based on rotation about the instantaneous angular velocity vector $\vec{\Omega}_{c.g.}$. For a finite rotation angle $\Delta\theta = |\vec{\Omega}_{c.g.}|\Delta t$, the final position of a vector $\vec{x}_r$ on the solid body with respect to $\vec{x}_{c.g.}$ can be expressed as (See Figure 10.91: Solid Body Rotation Coordinates (p. 741))

$$\vec{x}_r^{n+1} = \vec{x}_r^{n} + \Delta\vec{x} \tag{10.35}$$

where $\Delta\vec{x}$ can be shown to be

$$\Delta\vec{x} = |\vec{x}_r^{n} - \vec{x}_{c.g.}|\left[\sin(\Delta\theta)\hat{e}_\theta + (\cos(\Delta\theta) - 1)\,\hat{e}_r\right] \tag{10.36}$$

The unit vectors $\hat{e}_\theta$ and $\vec{e}_r$ are defined as

$$\hat{e}_\theta = \frac{\vec{\Omega}_{c.g.} \times \vec{x}_r}{|\vec{\Omega}_{c.g.} \times \vec{x}_r|} \tag{10.37}$$

$$\hat{e}_r = \frac{\hat{e}_\theta \times \vec{\Omega}_{c.g.}}{|\hat{e}_\theta \times \vec{\Omega}_{c.g.}|} \tag{10.38}$$

If the solid body is also translating with $\vec{v}_{c.g.}$, the $n+1$ position vector on the solid body can be expressed as

$$\vec{x}^{n+1} = \vec{x}_{c.g.}^{n} + \vec{v}_{c.g.}\Delta t + \vec{x}_r^{n+1} \tag{10.39}$$

where $\vec{x}_r^{n+1}$ is given by Equation 10.35 (p. 741).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.6.10. Previewing the Dynamic Mesh

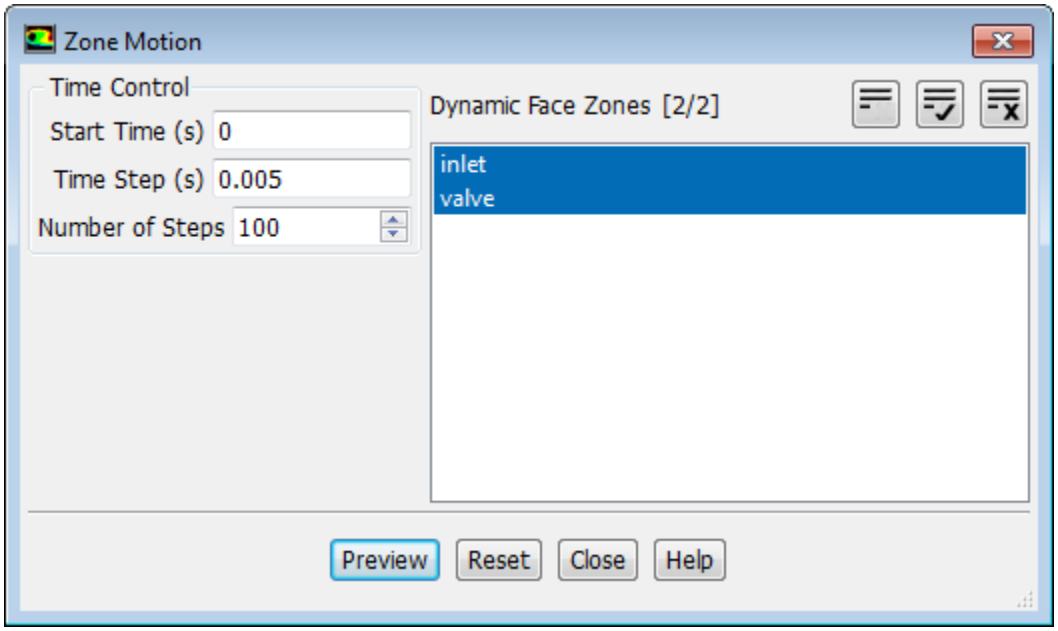
When you have specified the mesh update methods and their associated parameters, and you have defined the motion of dynamic zones, as described in Specifying the Motion of Dynamic Zones (p. 722), you can preview the motion of the mesh or the zone as it changes with time before you start your simulation. The same dynamic zone or mesh motion will be executed when you start your simulation.

10.6.10.1. Previewing Zone Motion

You can preview the motion of zones with **Rigid Body** or **User-Defined** motion using the Zone Motion Dialog Box (p. 2499) (Figure 10.92: The Zone Motion Dialog Box (p. 742)).

Setup → Dynamic Mesh → Display Zone Motion...

Figure 10.92: The Zone Motion Dialog Box



The zone motion preview only updates the graphical representation (in the graphics window) of the zones that you have selected in the **Dynamic Face Zones** list. Only zones that have been specified with either **User-Defined** or **Rigid Body** motion type are available for zone motion preview. To use the **Zone Motion** preview:

1. In the Zone Motion Dialog Box (p. 2499), select the face zones for which you want to preview the motion from the **Dynamic Face Zones** list. The **Dynamic Face Zones** list displays zones that have either **User-Defined** or **Rigid Body** motion specified. By default, all such zones are selected.

2. Enter the **Start Time**, **Time Step**, and **Number of Steps** under **Time Control**.

3. Click the **Preview** button to preview the zone motion. This positions the mesh according to the specified **Start Time**, and then integrates the position of the selected surfaces in time. The zone positions at the specified **Start Time** can be previewed without any subsequent motion by entering 0 for the **Number of Steps**.

4. Click **Reset** to restore the mesh to its initial state.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Previewing the zone motion can also be used as a postprocessor for six DOF simulations (see Six DOF Solver Settings (p. 705)).

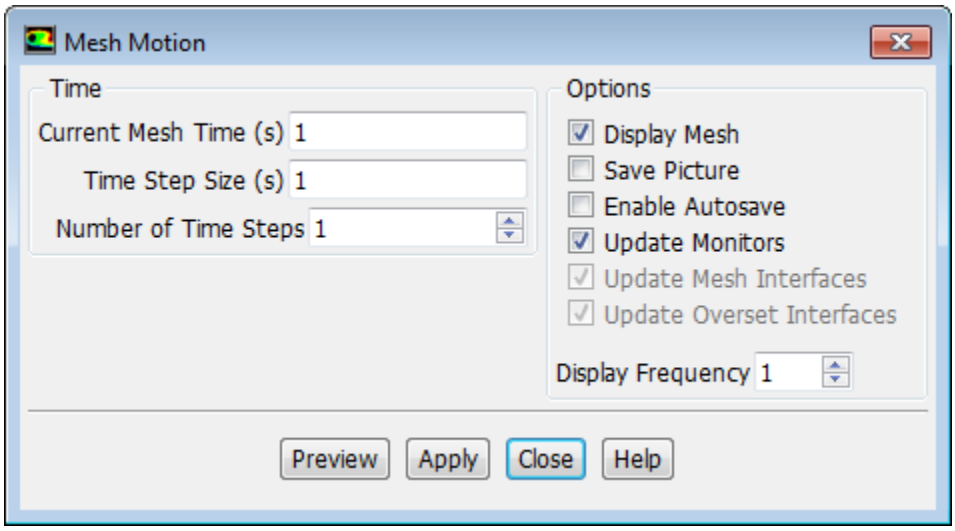
10.6.10.2. Previewing Mesh Motion

The mesh motion preview is different from the zone motion described above in that the mesh connectivity is changed in mesh motion.

To preview the dynamic mesh of a transient case, you can use the Mesh Motion Dialog Box (p. 2500) (Figure 10.93: The Mesh Motion Dialog Box (p. 743))

Setup → Dynamic Mesh → Preview Mesh Motion...

Figure 10.93: The Mesh Motion Dialog Box



The procedure is as follows:

1. Save the case file.

   File → Write → Case...

   Important

   Note that the mesh motion will actually update the node locations as well as the connectivity of the mesh, so you must be sure to save your case file before doing the dynamic mesh motion. Once you have advanced the mesh by a certain number of time steps, you will not be able to recover the previous status of the mesh, other than by reloading the appropriate ANSYS Fluent case file.

2. Specify the **Number of Time Steps** and the size of each time step (**Time Step Size**). The current time will be displayed in the **Current Mesh Time** field after the dynamic mesh has been advanced the specified number of steps.

   Note that if you turned on the in-cylinder option, the **Time Step Size** is automatically calculated from the **Crank Angle Step Size** and the **Crank Shaft Speed** that you have specified in the **In-Cylinder** tab of the Options Dialog Box (p. 2478).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. To view the dynamic mesh in the graphics window, enable the **Display Mesh** option. In addition, you can control the frequency at which ANSYS Fluent should display an updated mesh in the **Display Frequency** field. To save a picture file of the mesh each time ANSYS Fluent updates it during the preview, turn on the **Save Picture** option. This opens the Save Picture Dialog Box (p. 2568) (see Saving Picture Files (p. 102)).

4. Turn on **Enable Autosave** to use the automatic saving feature to specify the file name and frequency with which case and data files should be saved during the solution process. This opens the Autosave Case During Mesh Motion Preview Dialog Box (p. 2502).

   See Automatic Saving of Case and Data Files (p. 48) for details about the use of this feature. This provides a convenient way for you to save results at successive time steps for later postprocessing.

5. Enable the **Update Mesh Interfaces** option to update the interface at every time step.

6. Enable the **Update Overset Interfaces** option to update the overset interface at every time step.

7. Use the **Update Monitors** option to disable the processing of monitors and computation activities during mesh motion preview. This allows you to set up monitors and computational activities before running mesh motion preview without creating monitor files during the mesh motion preview.

8. Click **Preview** to start the preview. ANSYS Fluent will update the dynamic mesh by moving and deforming the face and cell zones that you have specified as dynamic zones. Click **Apply** to save your settings for mesh motion.

During the preview, information about the dynamic mesh will be displayed in the console window for each time step. Note that for the in-cylinder option, the reported `Maximum Cell Skew` is calculated only from zones undergoing remeshing. This ensures that you can always ascertain whether the skewness is increasing in the deforming zones. To report the maximum skewness of a cell from *any* zone, you can click the **Report Quality** button in the General Task Page (p. 2160).

Setup → **General** → **Report Quality**

10.6.11. Steady-State Dynamic Mesh Applications

While many dynamic mesh problems are transient, you can use dynamic meshes for steady-state applications as well. Some examples of steady-state applications include: checking the valve application after reaching a steady-state valve position; or after a fluid-structure interface application has reached a steady-state solution.

There are no differences in the meshing aspect between steady-state cases and transient cases. Furthermore, setting up a steady-state simulation is similar to setting up a transient case, described in Using Dynamic Meshes (p. 641). However, there are a few differences that you should note:

- A `CG_MOTION` UDF is needed to specify the motion of the boundary: a transient profile used in transient cases cannot be used in steady-state cases.
- The `dtime` passed to the `CG_MOTION` UDF is 1 by default: if a displacement of 1 mm is needed to move the boundary, you can specify the velocity to be 1e-3 m/s.
- Dynamic mesh parameters can be different since an interpolation error is no longer a concern.
- If you have enabled local remeshing for your steady-state application, you can instruct ANSYS Fluent to perform additional remeshing after the boundary has moved. This additional remeshing is based on skewness

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

criteria, and can further increase the quality of your mesh. See Dynamic Mesh Update Methods (p. 644) for further details.

The mesh must be manually updated through journal files or execute commands. To update the mesh, you can use the Mesh Motion Dialog Box (p. 2500).

Solution → Run Calculation → Update Dynamic Mesh...

Alternatively, you can use the following text command:

`solve → mesh-motion`

which can also be used as an execute command in the Execute Commands Dialog Box (p. 2535):

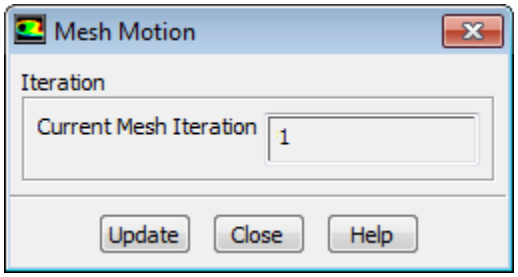
Note

When you are solving a problem using the six DOF solver, you may need to modify the pseudo time step size in order for the solver to converge. It is also recommended that you ensure that the steady-state solution is reasonably well converged before performing the steady-state dynamic mesh update.

Solution → Calculation Activities → Execute Commands → New...

You can display dynamic mesh statistics (such as minimum and maximum volumes and maximum cell and face skewness) by clicking the **Update** button in the Mesh Motion Dialog Box (p. 2500) (Figure 10.94: The Mesh Motion Dialog Box for Steady-State Dynamic Meshes (p. 745)).

Figure 10.94: The Mesh Motion Dialog Box for Steady-State Dynamic Meshes



Important

The following options are not available for steady-state applications:

- **In-Cylinder**
- **Implicit Update**

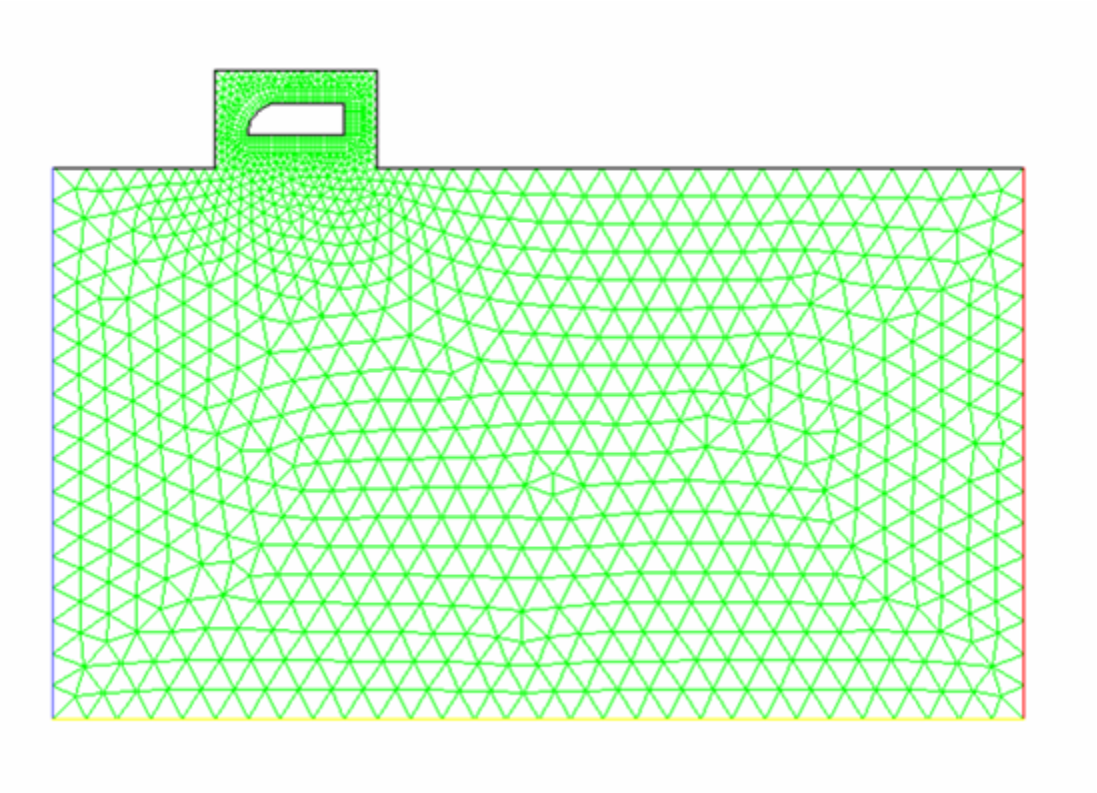
Also, the **Dynamic Mesh Events** dialog box is not available for steady-state applications.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.6.11.1. An Example of Steady-State Dynamic Mesh Usage

Consider a rescue drop case shown in Figure 10.95: Initial Object Position (p. 746). The object can be moved in any position in the steady-state solver, after which steady-state analyses can be performed at different object positions.

Figure 10.95: Initial Object Position



The dynamic mesh parameters setup is identical for the steady-state and transient cases, which is described in Setting Dynamic Mesh Modeling Parameters (p. 643). When setting up the dynamic zones, the procedures are similar to those described in Specifying the Motion of Dynamic Zones (p. 722), except that the UDF selected from the **Motion UDF/Profile** drop-down list is different. In steady-state cases the `dtime` passed to the UDF is by default 1. So, in this example, the object will move 50 mm each time the following UDF is executed:

```
#include "udf.h"

DEFINE_CG_MOTION(pod,dt,vel,omega,time,dtime)

{
 NV_S(vel,=,0);
 NV_S(omega,=,0);
 vel[1] = -50e-3;
}
```

The resulting mesh is shown in Figure 10.97: Final Object Position After 40 Executions (p. 747).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.96: The Mesh Motion Dialog Box After 40 Updates

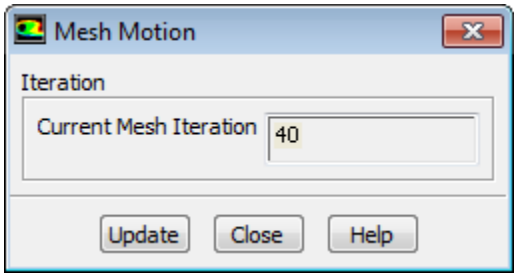
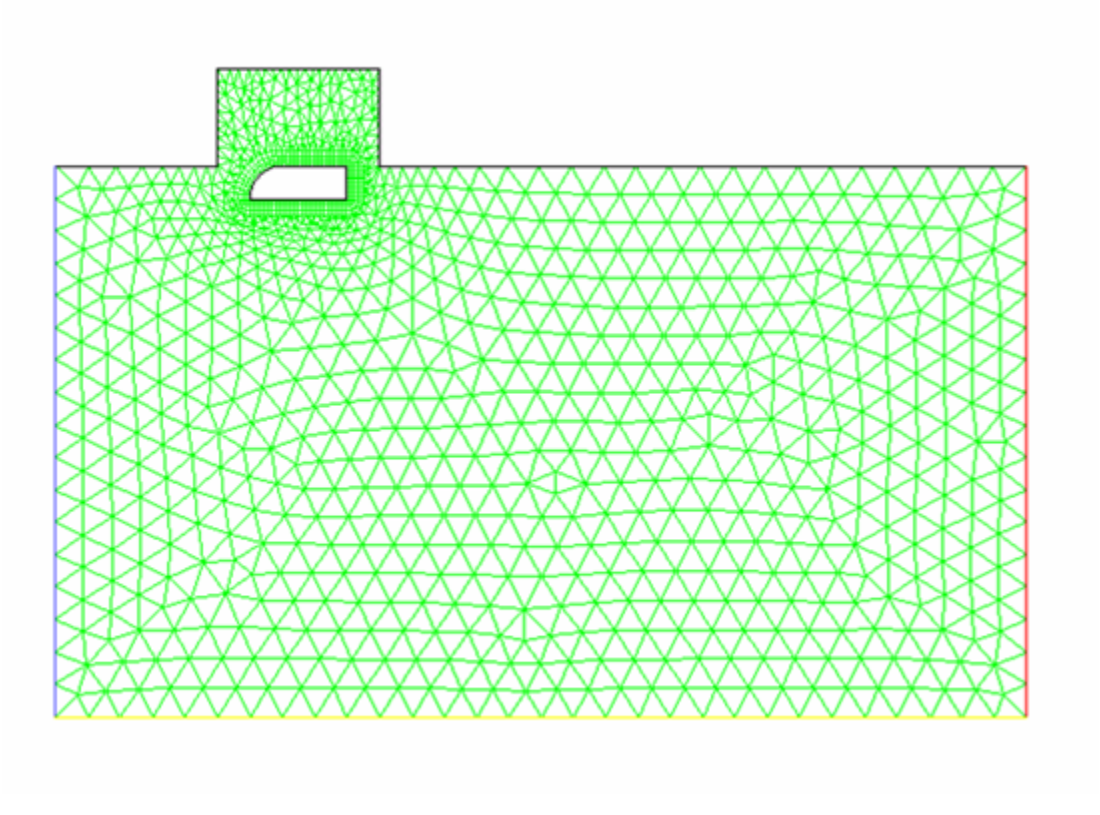


Figure 10.97: Final Object Position After 40 Executions



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 11: Modeling Turbulence

This chapter provides details about how to use the turbulence models available in ANSYS Fluent.

Information about turbulence modeling theory is presented in Turbulence in the Theory Guide. Information about using the turbulence models can be found in the following sections:

11.1. Introduction
11.2. Choosing a Turbulence Model
11.3. Steps in Using a Turbulence Model
11.4. Setting Up the Spalart-Allmaras Model
11.5. Setting Up the k-ε Model
11.6. Setting Up the k-ω Model
11.7. Setting Up the Transition k-kl-ω Model
11.8. Setting Up the Transition SST Model
11.9. Setting Up the Intermittency Transition Model
11.10. Setting Up the Reynolds Stress Model
11.11. Setting Up Scale-Adaptive Simulation (SAS) Modeling
11.12. Setting Up the Detached Eddy Simulation Model
11.13. Setting Up the Large Eddy Simulation Model
11.14. Model Constants
11.15. Setting Up the Embedded Large Eddy Simulation (ELES) Model
11.16. Setup Options for All Turbulence Modeling
11.17. Defining Turbulence Boundary Conditions
11.18. Providing an Initial Guess for k and ε (or k and ω)
11.19. Solution Strategies for Turbulent Flow Simulations
11.20. Postprocessing for Turbulent Flows

11.1. Introduction

Turbulence is the three-dimensional unsteady random motion observed in fluids at moderate to high Reynolds numbers. As technical flows are typically based on fluids of low viscosity, almost all technical flows are turbulent. Many quantities of technical interest depend on turbulence, including:

- Mixing of momentum, energy and species
- Heat transfer
- Pressure losses and efficiency
- Forces on aerodynamic bodies

While turbulence is, in principle, described by the Navier-Stokes equations, it is not feasible in most situations to resolve the wide range of scales in time and space by Direct Numerical Simulation (DNS) as the CPU requirements would by far exceed the available computing power for any foreseeable future. For this reason, averaging procedures have to be applied to the Navier-Stokes equations to filter out all, or at least, parts of the turbulent spectrum. The most widely applied averaging procedure is Reynolds-averaging (which, for all practical purposes is time-averaging) of the equations, resulting in the Reynolds-Averaged Navier-Stokes (RANS) equations. By this process, all turbulent structures are eliminated from

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the flow and a smooth variation of the averaged velocity and pressure fields can be obtained. However, the averaging process introduces additional unknown terms into the transport equations (Reynolds Stresses and Fluxes) that need to be provided by suitable turbulence models (turbulence closures). The quality of the simulation can depend crucially on the selected turbulence model and it is important to make the proper model choice as well as to provide a suitable numerical grid for the selected model. An alternative to RANS are Scale-Resolving Simulation (SRS) models. With SRS methods, at least a portion of the turbulent spectrum is resolved in at least a part of the flow domain. The most well-known such method is Large Eddy Simulation (LES), but many new hybrids (models between RANS and LES) are appearing. As all SRS methods require time-resolved simulations with relatively small time steps, it is important to understand that these methods are substantially more computationally expensive than RANS simulations.

ANSYS Fluent provides the following choices of turbulence models:

- Spalart-Allmaras model
- k-ε models
  - Standard k-ε model
  - Renormalization-group (RNG) k-ε model
  - Realizable k-ε model
- k-ω models
  - Standard k-ω model
  - Baseline (BSL) k-ω model
  - Shear-stress transport (SST) k-ω model
- v^2-f model (add-on)
- Transition k-kl-ω model
- Transition SST model
- Reynolds stress models (RSM)
  - Linear pressure-strain RSM
  - Quadratic pressure-strain RSM
  - Stress-Omega RSM
  - Stress-BSL RSM
- Scale-Adaptive Simulation (SAS) model, which can be used in combination with one of the following ω-based URANS models:
  - SST k-ω model
  - Standard k-ω model

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - BSL k-ω model
  - Transition SST model
  - ω-based Reynolds stress models (RSM)
- Detached eddy simulation (DES) model, which includes one of the following RANS models.
  - Spalart-Allmaras RANS model
  - Realizable k-ε RANS model
  - SST k-ω RANS model
  - BSL k-ω RANS model
  - Transition SST model
- Shielded Detached Eddy Simulation (SDES) model, which includes one of the following RANS models.
  - SST k-ω RANS model
  - BSL k-ω RANS model
  - Transition SST model
- Stress-Blended Eddy Simulation (SBES) model, which includes one of the following RANS models.
  - SST k-ω RANS model
  - BSL k-ω RANS model
  - Transition SST model
- Large eddy simulation (LES) model, which includes one of the following subgrid-scale models.
  - Smagorinsky-Lilly subgrid-scale model (with or without dynamic stress enabled)
  - WALE subgrid-scale model
  - Dynamic kinetic energy subgrid-scale model
  - Wall-Modeled LES (WMLES)
  - Wall-Modeled LES S-Ω (WMLES S-Ω)

11.2. Choosing a Turbulence Model

It is an unfortunate fact that no single turbulence model is universally accepted as being superior for all classes of problems. The choice of turbulence model will depend on considerations such as the physics of the flow, the established practice for a specific class of problem, the level of accuracy required, the available computational resources, and the amount of time available for the simulation. To make the most appropriate choice of model for your application, you need to understand the capabilities and limitations of the various options.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The purpose of this section is to give an overview of issues related to the turbulence models provided in ANSYS Fluent. The computational effort and cost in terms of CPU time and memory of the individual models is discussed. While it is impossible to state categorically which model is best for a specific application, general guidelines are presented to help you choose the appropriate turbulence model for the flow you want to model.

Note

For information on turbulence model compatibility with other Fluent models, see Appendix A: ANSYS Fluent Model Compatibility(p. 2847).

For additional information, see the following sections:

11.2.1. Reynolds Averaged Navier-Stokes (RANS) Turbulence Models
11.2.2. Scale-Resolving Simulation (SRS) Models
11.2.3. Grid Resolution SRS Models
11.2.4. Numerics Settings for SRS Models
11.2.5. Model Hierarchy

11.2.1. Reynolds Averaged Navier-Stokes (RANS) Turbulence Models

RANS models (Reynolds (Ensemble) Averaging in the Theory Guide) offer the most economic approach for computing complex turbulent industrial flows. Typical examples of such models are the k-ε or the k-ω models in their different forms. These models simplify the problem to the solution of two additional transport equations and introduce an Eddy-Viscosity (turbulent viscosity) to compute the Reynolds Stresses. More complex RANS models are available that solve an individual equation for each of the six independent Reynolds Stresses directly (Reynolds Stress Models – RSM) plus a scale equation (ε-equation or ω-equation). RANS models are suitable for many engineering applications and typically provide the level of accuracy required. Since none of the models is universal, you have to decide which model is the most suitable for a given applications.

11.2.1.1. Spalart-Allmaras One-Equation Model

The Spalart-Allmaras model is a relatively simple one-equation model that solves a modeled transport equation for the kinematic eddy (turbulent) viscosity. The Spalart-Allmaras model was designed specifically for aeronautics and aerospace applications involving wall-bounded flows and has been shown to give good results for boundary layers subjected to adverse pressure gradients. It is also gaining popularity in turbomachinery applications. Do not use the model as a general purpose model, as it is not well calibrated for free shear flows (large errors for example, jet flows).

The Spalart-Allmaras model has been extended within ANSYS Fluent with a y^+-insensitive wall treatment, which automatically blends all solution variables from their viscous sublayer formulation to the corresponding logarithmic layer values depending on y^+. The blending is calibrated to also cover intermediate y^+ values in the buffer layer $(1<y^+<30)$

11.2.1.2. k-ε Models

Two-equation models are historically the most widely used turbulence models in industrial CFD. They solve two transport equations and model the Reynolds Stresses using the Eddy Viscosity approach. The standard k-ε model in ANSYS Fluent falls within this class of models and has become the workhorse of practical engineering flow calculations in the time since it was proposed by Launder and Spalding [56] (p. 2890). Robustness, economy, and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow and heat transfer simulations.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The draw-back of some k-ε models is their insensitivity to adverse pressure gradients and boundary layer separation. They typically predict a delayed and reduced separation relative to observations. This can result in overly optimistic design evaluations for flows that separate from smooth surfaces (for example, aerodynamic bodies, diffusers). The k-ε model is therefore not widely used in external aerodynamics.

In ANSYS Fluent, the use of the Realizable k-ε model is recommended relative to other variants of the k-ε family. You should use the k-ε model in combination with either the Enhanced Wall Treatment (EWT-ε) or the Menter-Lechner near-wall treatment. For details, refer to Enhanced Wall Treatment ε-Equation (EWT-ε) or Menter-Lechner ε-Equation (ML-ε) in the Theory Guide, respectively. For cases where the flow separates under adverse pressure gradients from smooth surfaces (airfoils, and so on), k-ε models are generally not recommended.

11.2.1.3. k-ω Models

The ω-equation offers several advantages relative to the ε-equation. The most prominent one is that the equation can be integrated without additional terms through the viscous sublayer. This makes the formulation of a robust y^+-insensitive treatment relatively straightforward. Refer to y+-Insensitive Wall Treatment ω-Equation in the Theory Guide for details. Furthermore, k-ω models are typically better at predicting adverse pressure gradient boundary layer flows and separation. The downside of the standard ω-equation is a relatively strong sensitivity of the solution depending on the freestream values of k and ω outside the shear layer. The use of the standard k-ω model is, for this reason, not generally recommended in ANSYS Fluent.

The BSL and SST k-ω models have been designed to avoid the freestream sensitivity of the standard k-ω model, by combining elements of the ω-equation and the ε-equation. In addition, the SST model has been calibrated to accurately compute flow separation from smooth surfaces. Within the k-ω model family, it is therefore recommended to use either the BSL or SST model. These models are some of the most widely used models for aerodynamic flows. They are typically somewhat more accurate in predicting the details of the wall boundary layer characteristics than the Spalart-Allmaras model. The BSL and SST models (like all other ω-equation based models) use the y^+-insensitive wall treatment by default.

For the k-ω models, so-called low Reynolds number terms (low Re) have been proposed by Wilcox. These are available in ANSYS Fluent as an option. It is important to point out that these terms are not required for integrating the equations through the viscous sublayer. Their main influence lies in mimicking laminar-turbulent transition processes. However, this functionality is not widely calibrated and for wall-boundary layer transition, the combination of the SST model with the Transition SST model (Transition SST Model in the Theory Guide), Intermittency Transition model (Intermittency Transition Model in the Theory Guide), or the Transition k-kl-ω model (k-kl-ω Transition Model in the Theory Guide) is more reliable. The use of the low-Re terms is therefore not encouraged.

11.2.1.4. Reynold Stress Models

Reynolds Stress models (RSM) include several effects that are not easily handled by Eddy-Viscosity models. The most important effect is the stabilization of turbulence due to strong rotation and streamline curvature (as observed for example, in cyclone flows). RSM on the other hand often demand a significant increase in computing time partly due to the additional equations but mostly due to reduced convergence. This additional effort is not always justified by increased accuracy. Their usage is therefore not generally recommended and should be restricted to flows for which their superiority has been established, especially flow with strong swirl and rotation. If wall boundary layers are important, the combination of RSM with the ω- or BSL-equation is more accurate than the combination with the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ε-equation. The combination of RSM with BSL removes the free-stream sensitivity observed with the ω-equation in the same way as for two-equation models.

11.2.1.5. Laminar-Turbulent Transition Models

The following three models for transition prediction are available in ANSYS Fluent. Two of them can be combined with the scale-resolving methods described in Scale-Resolving Simulation (SRS) Models (p. 756).

- the Transition SST model (also known as the γ-Re_θ model)

  This model can be combined with Scale-Adaptive Simulation, Detached Eddy Simulation, Stress-Blended Eddy Simulation, and Shielded Detached Eddy Simulation.

- the Intermittency Transition model (also known as the γ model, and available for BSL, SST, Scale-Adaptive Simulation with BSL / SST, Detached Eddy Simulation with BSL / SST, Stress-Blended Eddy Simulation with BSL / SST, and Shielded Detached Eddy Simulation with BSL / SST)

- the Transition k-kl-ω model

For many test cases, the three models produce similar results. Due to their combination with the SST model, the Transition SST model and the Intermittency Transition model are recommended over the Transition k-kl-ω model. The Transition SST model is not Galilean invariant, and should therefore not be applied to surfaces that move relative to the coordinate system for which the velocity field is computed; furthermore, it is not suitable for fully developed pipe / channel flows where no freestream is present. For such cases, the Intermittency Transition model should be used instead (though you may need to adjust the Intermittency Transition model for pipe / channel flows by modifying the underlying correlations). Among the three models, only the Intermittency Transition model is capable of accounting for crossflow instability.

When using these models, note the following:

- These models are only applicable to wall-bounded flows. Like all other engineering transition models, they are not applicable to transition in free shear flows. They will predict free shear flows as fully turbulent.

- These models have not been calibrated in combination with other physical effects that affect the source terms of the turbulence model, such as:

  – buoyancy

  – multiphase turbulence

- No special calibration has been performed for the combination of the Transition SST and Intermittency Transition model with scale-resolving methods.

  Note that the activation of the LES term in the freestream in any hybrid RANS-LES model can affect the decay of the freestream turbulence, which in turn affects the transition location.

- Proper mesh refinement and specification of inlet turbulence levels is crucial for accurate transition prediction.

- In general, there is some additional effort required during the mesh generation phase because a low-Re mesh with sufficient streamwise resolution is needed to accurately resolve the transition region. Furthermore, in regions where laminar separation occurs, additional mesh refinement is necessary in order to properly capture the rapid transition due to the separation bubble.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The decay of turbulence from the inlet to the leading edge of the device should always be estimated before running a solution as this can have a large effect on the predicted transition location. Physically correct values for the turbulence intensity (Tu) should be achieved near the location of transition.

11.2.1.6. Curvature Correction for the Spalart-Allmaras and Two-Equation Models

One weakness of the eddy-viscosity models is that these models are insensitive to streamline curvature and system rotations, which play an important role in many turbulent flow applications. To sensitize the standard eddy-viscosity models to these curvature effects, you can use a modified turbulence production term. For more information, see Curvature Correction for the Spalart-Allmaras and Two-Equation Models in the *Fluent Theory Guide*.

11.2.1.7. Production Limiters for Two-Equation Models

A disadvantage of standard two-equation turbulence models is the excessive generation of the turbulence energy, G_k, in the vicinity of stagnation points. In order to avoid the buildup of turbulent kinetic energy in the stagnation regions, the production term in the turbulence equations can be limited by one of the two formulations described in Production Limiters for Two-Equation Models in the *Fluent Theory Guide*.

11.2.1.8. Model Enhancements

There are many model enhancements available for turbulence models. While such enhancements can improve simulations in some cases, they can also have detrimental effects. The general recommendation is therefore to use them with caution.

As noted above, the use of low-Re terms in combination with the ω-equation is not generally recommended.

Another model enhancement that should be used with care is the compressibility effects of Sarkar ([98] (p. 2892)). It can improve the prediction of free shear layers at high Mach numbers, but has also shown a pronounced negative impact on wall boundary layers. It is therefore not generally recommended. The compressibility effects option is available in the Viscous Model Dialog Box (p. 2176) when the compressible form of the ideal gas law or the real-gas model is enabled. This option is disabled by default (though this was not the case in version 14.5 and earlier) and it is recommended that you keep this option disabled for cases not involving free shear flows.

Buoyancy has a pronounced effect on turbulence (Effects of Buoyancy on Turbulence in the Theory Guide). The use of the source term in the k-equation is therefore recommended for buoyancy affected flows. The source term in the ε-equation and ω-equation is much less established and the term should be enabled with care.

11.2.1.9. Wall Treatment for RANS Models

It is recommended that you use a y^+-insensitive wall treatment for all models for which it is available (Spalart-Allmaras, ε-equation and ω-equation). It provides the most consistent wall shear stress and wall heat transfer predictions with the least sensitivity to y^+ values. (See an Overview of the Spalart-Allmaras model, Enhanced Wall Treatment ε-Equation (EWT-ε), Menter-Lechner ε-Equation (ML-ε), and y+-Insensitive Wall Treatment ω-Equation in the Theory Guide for more information.)

When Wall Functions are used, it is necessary to avoid grids with fine near wall spacing. It is recommended that you use $y^+>30$ in the entire domain. The application of Wall Functions is, however, not generally recommended as they do not allow a systematic refinement of the near wall grid. Wall Functions

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

are especially damaging for flows at low to medium Reynolds numbers (Re~10^4-10^6), as the assumption of an extended logarithmic layer is not valid in these cases. In case that Wall Functions are desired, the option of Scalable Wall Functions (Scalable Wall Functions in the Theory Guide) avoids the grid restrictions and can be run on fine meshes.

11.2.1.10. Grid Resolution for RANS Models

Grid generation has a strong impact on model accuracy. There are many considerations that have to be followed when generating high quality CFD grids. From a turbulence modeling standpoint, the most important one is that the relevant shear layers should be covered by at least ~10 cells normal to the layer. Below this resolution, the model will not be able to provide its calibrated performance. Especially for free shear flows, whose location is not known during grid generation, this is a requirement that is hard to achieve. Nevertheless, you should be aware that for lower resolution, the model performance can degrade.

For wall bounded flows, a structured mesh in wall-normal direction is highly recommended. The structured portion of the mesh should cover the entire boundary layer and extend beyond the boundary layer thickness to avoid restricting the growth of the boundary layer. Advanced turbulence models for wall boundary layers like the Spalart-Allmaras model and the SST model will only provide improved results to other models if a minimum of 10 or more structured (hex or prism) cells are located inside the boundary layer. In addition, one should ensure that the prism layer covers the wall boundary layer entirely. Note that these are not specific requirements of these models, but are general requirements for wall boundary layer simulations.

Both ε-based and ω-based models offer y^+-insensitive wall treatment options (see Enhanced Wall Treatment ε-Equation (EWT-ε), Menter-Lechner ε-Equation (ML-ε), and y+-Insensitive Wall Treatment ω-Equation in the Theory Guide for more information), which make the models relatively insensitive to the y^+-value of the wall cell. Generally speaking, it is more important to ensure that the boundary layer is covered with sufficient cells, then to achieve a certain y^+ criterion. However, for simulations with high accuracy demands on the wall boundary layer (especially for heat transfer predictions), near-wall meshes with y^+~1 are recommended. When wall functions are used, it is essential to avoid meshes with y^+ values lower than ~30 as the wall shear stress and the wall heat transfer can and will seriously deteriorate under such conditions. For this reason, the usage of y^+-insensitive wall treatments (to be selected for ε-equation and default for ω-equation based models) is recommended.

For transition models (see k-kl-ω Transition Model, Transition SST Model, and Intermittency Transition Model in the Theory Guide), more stringent grid resolution requirements apply than for standard RANS models, as transition modeling requires the resolution of the thin laminar boundary layer upstream of the transition location. For this reason, a low-Re mesh ($y^+<1$) with sufficient streamwise resolution is needed to accurately resolve the transition region. The expansion ratio of the grid normal to the wall should not exceed 1.1. Furthermore, in regions where laminar separation occurs, additional mesh refinement is necessary, in order to properly capture the rapid transition due to the separation bubble. Finally, the decay of turbulence from the inlet to the leading edge of the device should always be estimated before running a solution as this can have a large effect on the predicted transition location.

11.2.2. Scale-Resolving Simulation (SRS) Models

The alternative to RANS models are models that resolve at least a portion of the turbulence for at least a portion of the flow domain. Such models are generally termed 'Scale-Resolving'.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.2.2.1. Large Eddy Simulation (LES)

The most widely known SRS modeling concept is Large Eddy Simulation (LES). It is based on the approach of resolving large turbulent structures in space and time down to the grid limit everywhere in the flow. However, while widely used in the academic community, LES had very limited impact on industrial simulations. The reason lies in the excessively high resolution requirements for wall boundary layers. Near the wall, the largest scales in the turbulent spectrum are nevertheless geometrically very small and require a very find grid and a small time step. In addition, unlike RANS, the grid cannot only be refined in the wall normal direction, but also must resolve turbulence in the wall parallel plane. This can only be achieved for flows at very low Reynolds number and on very small geometric scales (the extent of the LES domain cannot be much larger than 10-100 times the boundary layer thickness parallel to the wall). For this reason the use of LES is only recommended for flows where wall boundary layers are not relevant and need not be resolved or for flows where the boundary layers are laminar due to the low Reynolds number. In such cases, the most balanced LES model is the WALE model (see Wall-Adapting Local Eddy-Viscosity (WALE) Model in the Theory Guide). It offers a good compromise between model complexity and generality. It also allows computing laminar shear (boundary) layers without any model impact.

An extension to LES is Wall-Modeled LES (WMLES). It allows the LES computation of wall bounded flows at higher Reynolds number without the large increase in grid resolution required for conventional LES at high Reynolds numbers. For WMLES, the grid resolution is largely independent of the Reynolds number with respect to the grid cells required per boundary layer volume. An enhanced version of WMLES called WMLES S-Ω is also available. WMLES does not provide zero eddy-viscosity for flows with constant shear. Therefore, it does not allow the computation of transitional effects, and can produce overly large eddy-viscosities in separating shear layers. The enhanced WMLES S-Ω formulation overcomes these deficiencies. See Algebraic Wall-Modeled LES Model (WMLES) in the Theory Guide for further details.

11.2.2.2. Hybrid RANS-LES Models

In order to avoid the high resolution requirements of LES, numerous hybrid models have been developed in recent years. These are models that combine certain elements of RANS and LES approaches in a way that allows for the simulation of high Reynolds number flows. With hybrid models, the attached wall boundary layers are typically covered by the RANS part of the model, while large detached regions are handled in 'LES' mode, meaning with a partial resolution of the turbulent spectrum in space and time. Hybrid models rely on a strong enough flow instability to generate turbulent structures in the separated zone. This is typically the case for flows behind bluff bodies, where URANS models predict single-mode periodic vortex shedding. Hybrid models allow these vortices to generate smaller eddies down to the available grid limit. Figure 11.1: Illustration of SST-URANS vs. SST-SAS Models (p. 758) shows a typical scenario: While the application of a standard RANS model in unsteady mode results in a single frequency vortex shedding (left), the application of hybrid models allows a break-up of the large structures into smaller scales. This is beneficial for predicting the correct mixing behind the body or to extract spectral information (for example, for acoustic simulations). At the same time, the wall boundary layers are covered by the RANS part of the hybrid model avoiding the excessive resolution requirements of LES.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.1: Illustration of SST-URANS vs. SST-SAS Models

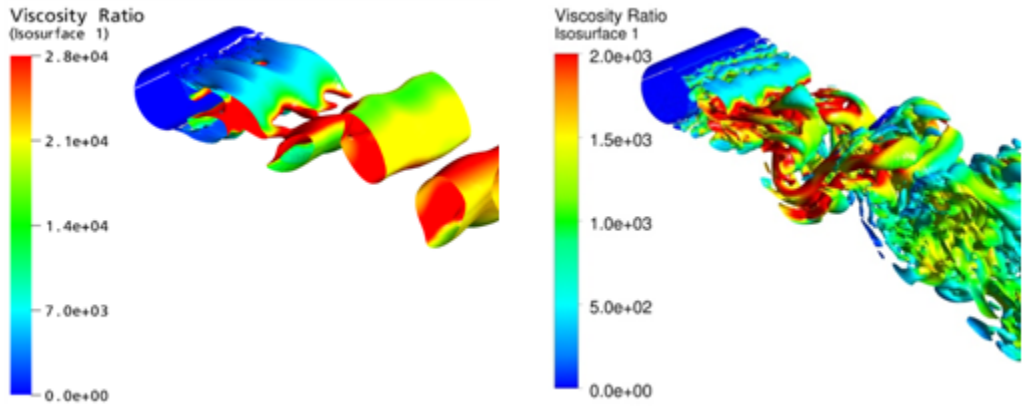


Figure 11.1: Illustration of SST-URANS vs. SST-SAS Models (p. 758) shows a circular cylinder in a cross flow at $\mathrm{Re}=3.6\times10^6$. The left hand side illustrates the SST-URANS model, while the right-hand side illustrates the SST-SAS model. The isosurface of $\mathrm{Q}=\mathrm{S}^2-\Omega^2$ is colored according to the eddy viscosity ratio μ_t/μ (note that the scale in the right-hand side figure is smaller by a factor of 14).

11.2.2.2.1. Scale-Adaptive Simulation (SAS)

The SAS modeling approach (see Scale-Adaptive Simulation (SAS) Model in the Theory Guide) as proposed by Menter et al. ([69] (p. 2890) [70] (p. 2890)) is based on the introduction of the von Karman length scale, L_{vK}, into the turbulence equations (for the BSL and SST models, it enters into the ω-equation). L_{vK} is defined as the ratio of the first divided by the second derivative of the velocity vector (times the von Karman constant κ=0.41):

$$L_{vK}=\kappa\left|\frac{U'}{U''}\right|; \qquad U''=\sqrt{\frac{\partial^2 U_i}{\partial x_k^2}\frac{\partial^2 U_i}{\partial x_j^2}}\ ;$$
$$U'=S=\sqrt{2\cdot S_{ij}S_{ij}}\ ; \quad S_{ij}=\frac{1}{2}\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right) \tag{11.1}$$

The inclusion of this term allows the model to adjust its length scale to already resolved scales in the flow and thereby provide a low enough eddy viscosity to allow the model to operate in 'LES' mode.

The SAS approach has the advantage that the RANS part of the model is unaffected by the grid spacing and will therefore not allow a deterioration of model accuracy as seen in DES in regions of refined grid but insufficient flow instability. However, in cases where the flow instability is not strong enough, the SAS will remain in RANS mode and will not produce unsteady structures. While this is often a sign that the RANS model is still reasonably capable of handling the flow, it is a limitation if unsteady information is required (for example, in acoustics). In such cases, the internal interface option of the ELES implementation can be applied to convert modeled turbulence into resolved turbulence (see Internal Interface Without LES Zone in the Theory Guide).

11.2.2.2.2. Detached Eddy Simulation (DES)

The DES model (see Detached Eddy Simulation (DES) in the Theory Guide) achieves the switch between RANS and LES by a comparison of the turbulent length scale L_t with the grid spacing $\Delta_{\max}$. The model selects the minimum of both and thereby switches between RANS and LES mode by replacing ε in the k-equation by:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\varepsilon = \frac{k^{3/2}}{L_t} \rightarrow \varepsilon = \frac{k^{3/2}}{\min\left(L_t, c_{DES}\Delta_{\max}\right)} \tag{11.2}$$

Once the model selects the grid spacing as the minimum, the model is operating in 'LES' mode.

The grid spacing enters explicitly into the DES model. This can affect the RANS solution in regions, where the grid is between RANS and LES resolution (so-called 'gray zones' in DES) and/or where the flow instability is not strong enough to generate LES structures. Another issue to consider with DES is the problem of 'grid-induced-separation' (GIS). It occurs if the grid for an attached wall boundary layer flow is refined to a point where the DES limiter becomes active and affects the RANS solution. However, in such situations, the flow instability is not strong enough to balance the reduced RANS content by resolved turbulence. This will typically result in an artificial flow separation at the location of grid refinement. It typically happens if $\Delta_{\max} < \delta$ (δ being the boundary layer thickness). Remedies for this situation have been proposed by Menter et al. who recommended using the F1 blending function of the SST-DES model to shield the boundary layers from the DES limiter. Later, alternative blending functions for the same purpose have been proposed by Spalart et al. ([99] (p. 2892))—resulting in the terminology Delayed DES (DDES). The DDES model as originally proposed for the Spalart-Allmaras model provided limited protection against GIS for two-equation models such as BSL, SST, and k-ε. Therefore, the DDES function has been re-calibrated for the BSL, SST, and k-ε models and is now the recommended choice and the default setting when using these models.

A further refinement is provided by the Improved DDES (IDDES) formulation of Strelets et al. ([105] (p. 2892)), which extends the LES zone of the model to the outer part of wall boundary layers. This allows the simulation of wall boundary layers in Wall-Modeled LES (WMLES) mode. In this model, the IDDES model is applied like a LES model, typically with the specification of unsteady inlet conditions. The grid resolution requirements for WMLES are much less stringent than for LES.

All of the above shielding function variants are available in ANSYS Fluent. For the Spalart-Allmaras model, the original DDES shielding function is used. For the k-ε, BSL, and SST models, the DDES function has been re-calibrated for better boundary layer protection. The re-calibrated DDES function is the default selection and is recommended over the use of the F1 or F2 functions for the BSL / SST model. Despite the potential difficulties in the application of hybrid methods, they have the potential to greatly expand the usage of Scale-Resolving Simulation models for engineering applications, as they avoid the excessive resolution required by LES for wall boundary layers. IDDES in WMLES mode is an advanced option and should only be used if you are familiar with the original literature and the grid requirements for this model.

The DES and DDES models in ANSYS Fluent have been superseded by the SDES and SBES models described in Shielded Detached Eddy Simulation (SDES) and Stress-Blended Eddy Simulation (SBES) (p. 759).

11.2.2.2.3. Shielded Detached Eddy Simulation (SDES) and Stress-Blended Eddy Simulation (SBES)

The Shielded Detached Eddy Simulation (SDES) model and the Stress-Blended Eddy Simulation (SBES) model are hybrid RANS-LES models with an improved shielding function compared to DDES / IDDES. They provide strong shielding of the RANS boundary layer, faster "transition" from RANS to LES in separating shear layers, and (for SBES) explicit selection of the LES model. For implementation details, see Shielded Detached Eddy Simulation (SDES) and Stress-Blended Eddy Simulation (SBES) in the Theory Guide; for setup details, see Including the SDES or SBES Model with BSL, SST, and Transition SST Models (p. 805).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These models are recommended over the older DES / DDES model formulations. The SBES model is considered the optimal selection, as it offers the clearest distinction between RANS and LES zones.

11.2.2.3. Zonal Modeling and Embedded LES (ELES)

As pointed out in the previous sections, hybrid models rely on flow instabilities to generate turbulent structures in large separated regions without the explicit introduction of unsteadiness through the boundary conditions. However, there are situations, where such instabilities are not present or are not reliable to serve this purpose. In such cases, it is desirable to apply RANS and the LES models in pre-defined zones and provide clearly defined interfaces between them. At these interfaces, the modeled turbulent kinetic energy from the upstream RANS model is converted explicitly to resolved scales at an internal boundary to the LES zone. The LES zone can then be limited to the region of interest where unsteady results are required. ELES is available in ANSYS Fluent and allows the combination of most RANS model with classical LES models. It is important to emphasize that in this mode, a full LES resolution is required within the LES zone. In the LES zone, using the WALE model is recommended (see Wall-Adapting Local Eddy-Viscosity (WALE) Model in the Theory Guide).

11.2.3. Grid Resolution SRS Models

Grid Resolution SRS models are discussed in the following sections:

11.2.3.1. Wall Boundary Layers
11.2.3.2. Free Shear Flows

11.2.3.1. Wall Boundary Layers

For wall boundary layers, it is important to distinguish if they are computed in RANS or wall-resolved or Wall-Modeled LES (WMLES) mode. Only the IDDES and the SBES model can be run in WMLES mode reliably.

In the case of RANS mode, the requirements are the same as for any RANS model. For a wall-resolved LES, it is typically recommended to use a mesh with a grid spacing scaling with $\Delta x^+ \approx 40, \Delta y^+ \approx 1, \Delta z^+ \approx 20$ where x is the streamwise, y the wall normal and z the spanwise direction (for example, channel flow). However, in complex applications, the distinction between streamwise and spanwise direction is not feasible and then $\Delta x^+ \approx 20, \Delta y^+ \approx 1, \Delta z^+ \approx 20$ would be required. This scaling demonstrates the strong Reynolds number dependency of the LES approach for wall bounded flows.

For the IDDES and SBES models in WMLES mode, the above requirements can be relaxed. The grid spacing no longer scale with the wall friction, but with the boundary layer thickness δ. It is recommended that you use $\Delta x \approx \Delta z \approx (0.05-0.1)\cdot\delta$. The wall normal resolution should be like for a finely resolved RANS simulation, meaning a near wall resolution of $\Delta y^+ \approx 1$ and around 30 nodes inside the boundary layer.

11.2.3.2. Free Shear Flows

For free shear flows, it is difficult to provide general recommendations, as there are many different flow scenarios. The current recommendation is therefore based on the most common (and most frequent) free shear flow – a turbulent mixing layer. It will not necessarily apply to other free shear flows like jets and you are advised to perform tests as to the optimal resolution of your specific flow.

For free shear flows and SRS models, one should aim for uniform isotropic cells (all edges have similar length). The shear layer should be covered by ~10-20 cells. As the layer thickness is often hard to estimate, one can also look at the ratio

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$R_l = \frac{L_t}{\Delta_{\max}}; \quad L_t = \frac{k^{3/2}}{\varepsilon} = \frac{k^{1/2}}{0.09\omega} \tag{11.3}$$

Where $\Delta_{\max}$ is the maximum cell edge length. It is available in ANSYS CFD-Post for all turbulence models. This estimate should be based on the RANS solution and *not* on the SAS / DES / SDES / SBES solution for the given flow. The above requirement for the mixing layer translates into:

$$R_l > 5-10 \tag{11.4}$$

In these estimates, 10 cells across the layer (or $R_l = 5$) is truly a lower limit. The higher values of 20 cells and $R_l = 10$ are much safer.

11.2.4. Numerics Settings for SRS Models

For Scale-Resolving Simulations (SRS), specific discretization and solver settings are required to achieve optimal accuracy with minimal numerical effort. The recommendations given below should be considered as a starting point for your specific flow application and are not generic, but problem dependent. The recommendations are based on incompressible, single phase flow without chemical reactions or other complex additional physics. In case your simulation features additional complex physical effects, it requires adjusting the recommended solver settings accordingly. In most cases, this will mean that a higher effort must be invested into the coupling of the equations (for example lower time step, reduced under-relaxation, higher iteration count, smaller residuals), in order to avoid a de-coupling of different physical phenomena.

For SRS models, it is generally recommended to use the pressure-based solver, as it offers optimized schemes for resolving turbulence relative to the density based solver.

For Scale-Resolving Simulations, optimal numerics settings are essential for achieving accurate results in an acceptable time frame. The reason is that at the SRS models operate at the resolution limit of the provided grid where the smallest scales are of the order of the grid spacing and the time resolution. Numerics settings therefore have to be chosen to provide an optimal balance between accuracy and robustness.

It is generally recommended to initialize the solution from a (reasonably) converged RANS simulation.

11.2.4.1. Time Discretization

For Scale-Resolving Simulations, the resolution of the turbulent structures in time is essential for the success of the simulation. This is, to the largest extent, defined by the selected time step. As the SRS model is operating at the grid limit, you should select a time step that ensures a Courant-Friedrichs-Levy (CFL) number of

$$CFL = \frac{U\Delta t}{\Delta x} < 1 \tag{11.5}$$

The CFL number is computed by the solver and can be checked based on an initial RANS simulation, for example. It is important to emphasize that this is not a numerical limit and that the solver will be able to sustain much larger CFL numbers. In complex applications, there will always be limited regions of fine cells or large velocities and you should not restrict the CFL number based on such zones. The recommendation of CFL =1 should be applied in the main SRS region in combination with a uniform isotropic grid. It is recommended that you vary the time step for each type of application and explore its optimal value. This can substantially save on computing costs.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The time derivative should be computed by the Second Order Implicit option. (See User Inputs for Time-Dependent Problems (p. 1609) for guidelines on setting solution parameters for transient calculations in general.)

11.2.4.2. Spatial Discretization

For SRS models, it is important to minimize the numerical dissipation of the scheme in order to avoid damping of the smallest scales by numerical dissipation. For spatial discretization, the choice is between the Central Differencing (CD) scheme (see Central-Differencing Scheme in the Theory Guide) and the Bounded Central Differencing scheme (BCD) (see Bounded Central Differencing Scheme in the Theory Guide). The Central Differencing scheme is the least dissipative and provides the highest resolution accuracy for the smallest scales. Especially for aero-acoustics simulations, where the spectral content at higher frequencies can be important, this is a desirable feature. However, Central Differencing schemes are prone to solution oscillations (checker boarding) in the velocity field. When using the Central Differencing scheme, it is therefore important to provide a high quality mesh (no mesh jumps, isotropic cells and high resolution in critical zones) and to avoid large time steps (the CFL number should be smaller than 1 in the main SRS region). It is recommended that you visually monitor the solution regularly in order to avoid wasting computational resources. In case oscillations appear, the choice is to: improve the mesh; reduce the time step; or to switch to the slightly more dissipative, but also more robust Bounded Central Differencing scheme. In many complex applications, the Bounded Central Differencing Scheme is the numerics option of choice. It typically provides sufficiently low dissipation to allow the turbulent structures to evolve, but, at the same time, is robust enough to handle non-optimal grids as they are typically encountered in industrial simulations. In addition, the Bounded Central Differencing scheme is also suitable for hybrid methods like SAS, DES, SDES, and SBES, and will provide stable solutions in RANS regions, with highly stretched grids and with a CFL number larger than 1. For ELES, the numerical scheme in the LES zone can be selected independently from the settings in the RANS zone. The RANS region can then be computed with standard higher-order upwind schemes and the LES zone can be covered by either the Central Differencing scheme or the Bounded Central Differencing scheme.

For the gradient calculation, it is recommended that you select the Least Squares Cell Based option, or the Green-Gauss Node Based option, to ensure a second-order interpolation on non-orthogonal grids.

For pressure interpolation, it is recommended that you use the second-order scheme, or the body-force-weighted scheme. Due to its higher dissipation, the PRESTO! scheme (see Pressure Interpolation Schemes in the Theory Guide) can result in a delayed formation or damping of turbulent structures. It is therefore not recommended for SRS.

11.2.4.3. Iterative Scheme

The selection of the iterative scheme will mostly affect the computational costs as the cost per iteration between these methods is rather different. However, recommendations are not straightforward, as the higher cost per iteration of a scheme can be offset by faster convergence within the time step.

The fastest scheme is the Non-Iterative Time Advancement (NITA) scheme (see Non-Iterative Time-Advancement Scheme in the Theory Guide as well as Setting Solution Controls for the Non-Iterative Solver (p. 1566)). This scheme typically works well for limited LES zones and high quality meshes. It is also important to ensure a low time step of a CFL number below 1. For the NITA scheme, all explicit under-relaxation factors are, by default, set equal to one. With NITA, it has slight advantages to use the fractional step method if there is no involvement of more complex physics, when otherwise the PISO scheme can be more beneficial.

In case the application is too complex for the NITA scheme to provide a solution, the iterative SIMPLEC (SIMPLE vs. SIMPLEC (p. 1562)) or PISO (PISO (p. 1562)) schemes are the next possible option. They should

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

be preferred relative to the SIMPLE scheme, as they show faster convergence per time step and can be run with more aggressive under-relaxation (higher values equal to 1 or close to 1). If such settings prevent convergence within the time step, then check if your time step is small enough for maintaining CFL numbers below 1 in the SRS region – if not, then try reducing Δt. If this is not possible, or does not lead to satisfactory convergence, then reduce the under-relaxation factors to values between the default settings and 1. For skewed meshes or meshes with problematic quality, the reduction of explicit relaxation factor for pressure correction down to 0.7 from 1 can be very helpful.

11.2.4.3.1. Convergence Control

The convergence criterion within each time step will strongly affect solution costs, as a low criterion will result in an increased number of iterations. It is not possible to provide general recommendations, as the required residual depends on the application. The most relevant residual in the SRS models is the continuity residual. It should be converged by approximately 2 orders of magnitude per time step for a CFL number ~ 1. This should be achievable with around 5-10 maximum iterations per time step for flows involving no other physical models. Be sure to check the impact of convergence on your solution to ensure that the residual is reduced to a level consistent with your problem.

For simulations with small CFL values, the **Extrapolate variables** option (available in the Run Calculation Task Page (p. 2537)) can be very beneficial. This can substantially reduce the number of iterations within the time step up to 40% when the convergence control is based on the convergence criteria.

In case your simulation requires the combination of additional physical models, such as combustion or multiphase, a reduction of the under-relaxation factors might be required. In this case, the number of maximum iterations per time step can be increased to achieve the desired residual reduction.

For hybrid RANS/LES simulations, such as the SAS, DES, SDES, and SBES models, there can be situations where the RANS portion of the flow limits convergence (for example, due to poor grid quality). In such cases, the application of the coupled solver should be considered. For the coupled pressure-based solver, each iteration is typically more expensive than for the SIMPLEC algorithm, but this is at least partly offset by faster convergence. The recommendations concerning residuals are similar to the SIMPLEC method, but the maximal number of iterations can be reduced to values as low as 2-5 for highly unstable flows, and 5-10 for more sensible flows and acoustics simulations.

11.2.5. Model Hierarchy

As discussed, turbulence modeling is a balance between accuracy and cost. The recommendation is to use RANS models as much as possible and as long as they provide the accuracy required for the simulation. RANS models will remain the workhorse of turbulence modeling for many years to come. Within the RANS family, eddy-viscosity models are typically sufficient for most engineering flow simulations. The application of RSM is only recommended for flows that are known to systematically benefit from their usage and justify the increase in computing power.

In cases where steady RANS or URANS models cannot provide the accuracy or unsteady information required, it is recommended that you switch to the SAS approach. It is relatively forgiving in terms of the grid resolution and will not deteriorate the results in case of insufficient resolution in the unsteady zone. The SAS model will only provide scale-resolution if a strong flow instability is present. Visual inspection (using isosurfaces of the Q criterion) will quickly allow a judgment if the model provides sufficient unsteadiness and resolution relative to the grid spacing. In such cases, the internal interface option of the ELES implementation can be applied to convert modeled turbulence into resolved turbulence (see Internal Interface Without LES Zone in the Theory Guide).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DDES (DES is not recommended) models can allow the formation of unsteadiness even for cases where SAS remains stable. DDES does require a more carefully crafted grid in the LES zone due to the DES grid influence on the RANS solution. For the DES-BSL / SST model, use the default DDES blending function for shielding.

SDES and SBES have an improved shielding function compared to DDES / IDDES. They provide strong shielding of the RANS boundary layer and demonstrate a fast "transition" between RANS and LES in separating shear layers. The SBES model formulation in combination with the WALE LES model is recommended over older models like DES, DDES, and IDDES.

ELES is recommended in cases where limited zones with high accuracy requirements are embedded inside a larger RANS zone. At the interface between the RANS and the LES zone, unsteady turbulence is generated either by the Vortex Method or the Spectral Synthesizer. For wall bounded flows, the ELES method requires a very fine near wall resolution in the LES zone.

Pure LES should only be applied to free shear flows (for example, combustion chambers without wall influence) or to very limited domains using meshes with fine LES near wall resolution. It is important to be aware of the strong increase of grid resolution requirements with Reynolds number for wall bounded flows. For wall boundary layers at higher Reynolds numbers, the IDDES model can be run in WMLES mode – meaning with specified unsteady inlet conditions.

11.3. Steps in Using a Turbulence Model

When your ANSYS Fluent model includes turbulence you need to enable the relevant model and options, and supply turbulent boundary conditions. These inputs are described in this section.

The procedure for setting up a turbulent flow problem is described below. (Note that this procedure includes only those steps necessary for the turbulence model itself; you will need to define the other settings—for example, other models, boundary conditions—as usual.)

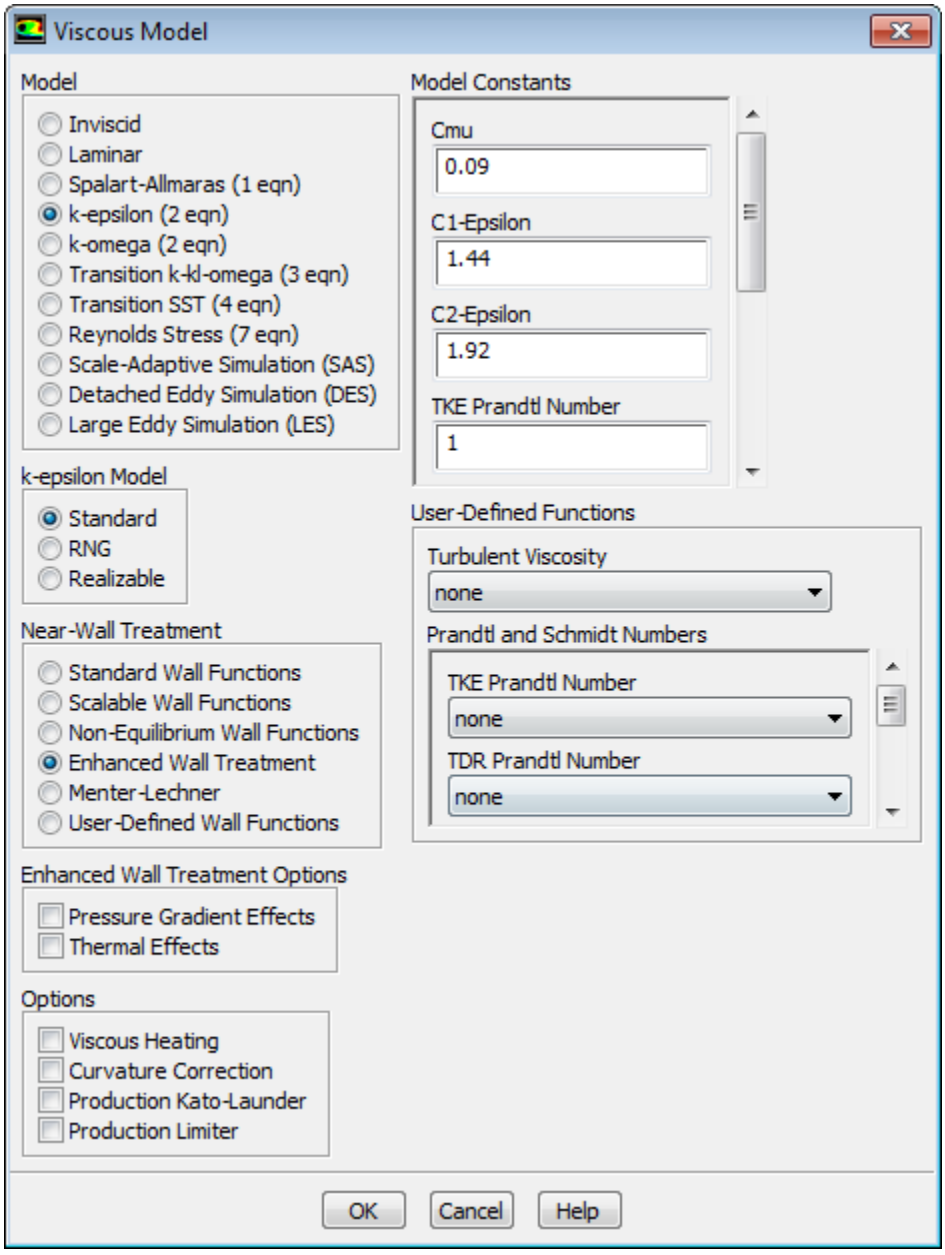
1. Define the turbulence model settings.

   a. Open the Viscous Model Dialog Box (p. 2176) (Figure 11.2: The Viscous Model Dialog Box (p. 765)) by right-clicking **Viscous** in the tree (under **Setup/Models**) and clicking **Edit...** in the menu that opens.

      Setup → **Models** → **Viscous** → **Edit...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.2: The Viscous Model Dialog Box



b. Select either **Spalart-Allmaras**, **k-epsilon**, **k-omega**, **Transition k-kl-omega**, **Transition SST**, **Reynolds Stress**, **Scale-Adaptive Simulation (SAS)**, **Detached Eddy Simulation (DES)**, or **Large Eddy Simulation (LES)** from the **Model** list.

If you choose the **k-epsilon** model, select either **Standard**, **RNG**, or **Realizable** from the **k-epsilon Model** list. If you choose the **k-omega** model, select **Standard**, **BSL**, or **SST** from the **k-omega Model** list.

Important

The **Large Eddy Simulation (LES)** model is available only for 3D cases.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. If the flow involves walls, and you are using one of the k-ε models or the ε-based Reynolds stress models, choose one of the following options from the **Near-Wall Treatment** list in the **Viscous Model** dialog box:

- **Standard Wall Functions**
- **Scalable Wall Functions**
- **Non-Equilibrium Wall Functions**
- **Enhanced Wall Treatment**
- **Menter-Lechner** (for k-ε models only)
- **User-Defined Wall Functions**

For more information about these near-wall options, see Near-Wall Treatments for Wall-Bounded Turbulent Flows in the Theory Guide. By default, the standard wall function is enabled.

For more information about the automatically defined near-wall treatment for the Spalart-Allmaras model, see Wall Boundary Conditions in the Theory Guide.

For more information about the automatically defined near-wall treatment for the k-ω model, see Wall Boundary Conditions in the Theory Guide.

For more information about the automatically defined near-wall treatment and the optional Werner-Wengle near-wall treatment for the LES model, see Inlet Boundary Conditions for the LES Model and LES Near-Wall Treatment in the Theory Guide, respectively.

d. Enable the other appropriate turbulence modeling options in the **Viscous Model** dialog box, as appropriate. See Setup Options for All Turbulence Modeling (p. 797)

2. Specify the boundary conditions for the solution variables in the appropriate boundary condition dialog boxes. The boundary condition dialog boxes can be opened by right-clicking the boundary name in the tree (under **Setup/Boundary Conditions**) and clicking **Edit...** in the menu that opens; alternatively, you can open them from the **Boundary Conditions** task page:

Setup → **Boundary Conditions**

See Defining Turbulence Boundary Conditions (p. 807) for details.

3. Specify the initial guess for the solution variables in the **Solution Initialization** task page.

Solution → **Initialization**

See Providing an Initial Guess for k and ε (or k and ω) (p. 810) for details. Note that Reynolds stresses are automatically initialized using k, and therefore need not be initialized explicitly.

11.4. Setting Up the Spalart-Allmaras Model

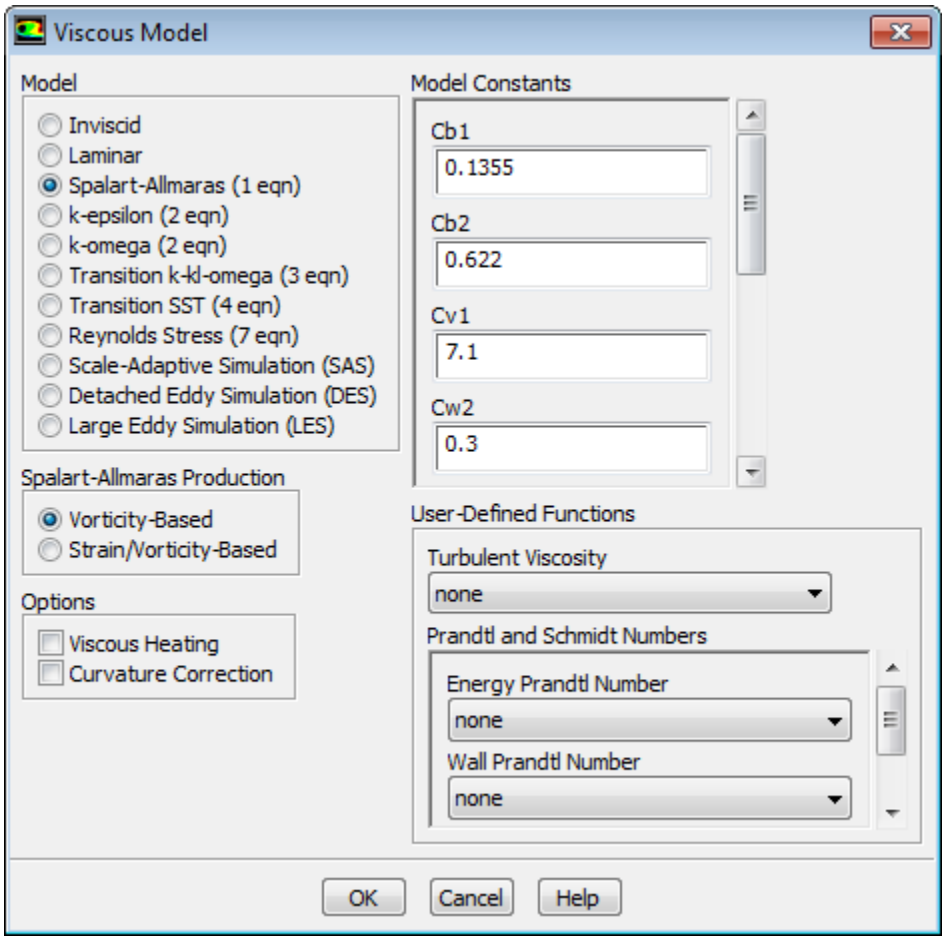
If you choose the Spalart-Allmaras model, the following options are available:

- vorticity-based production (Vorticity- and Strain/Vorticity-Based Production (p. 800))

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- strain/vorticity-based production (Vorticity- and Strain/Vorticity-Based Production (p. 800))
- viscous heating (always enabled for the density-based solvers) (Including the Viscous Heating Effects (p. 798))
- curvature correction (Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models (p. 798))

Figure 11.3: The Viscous Model Dialog Box Displaying the Spalart-Allmaras Production



11.5. Setting Up the k-ε Model

For additional information, see the following sections:
11.5.1. Setting Up the Standard or Realizable k-ε Model
11.5.2. Setting Up the RNG k-ε Model

11.5.1. Setting Up the Standard or Realizable k-ε Model

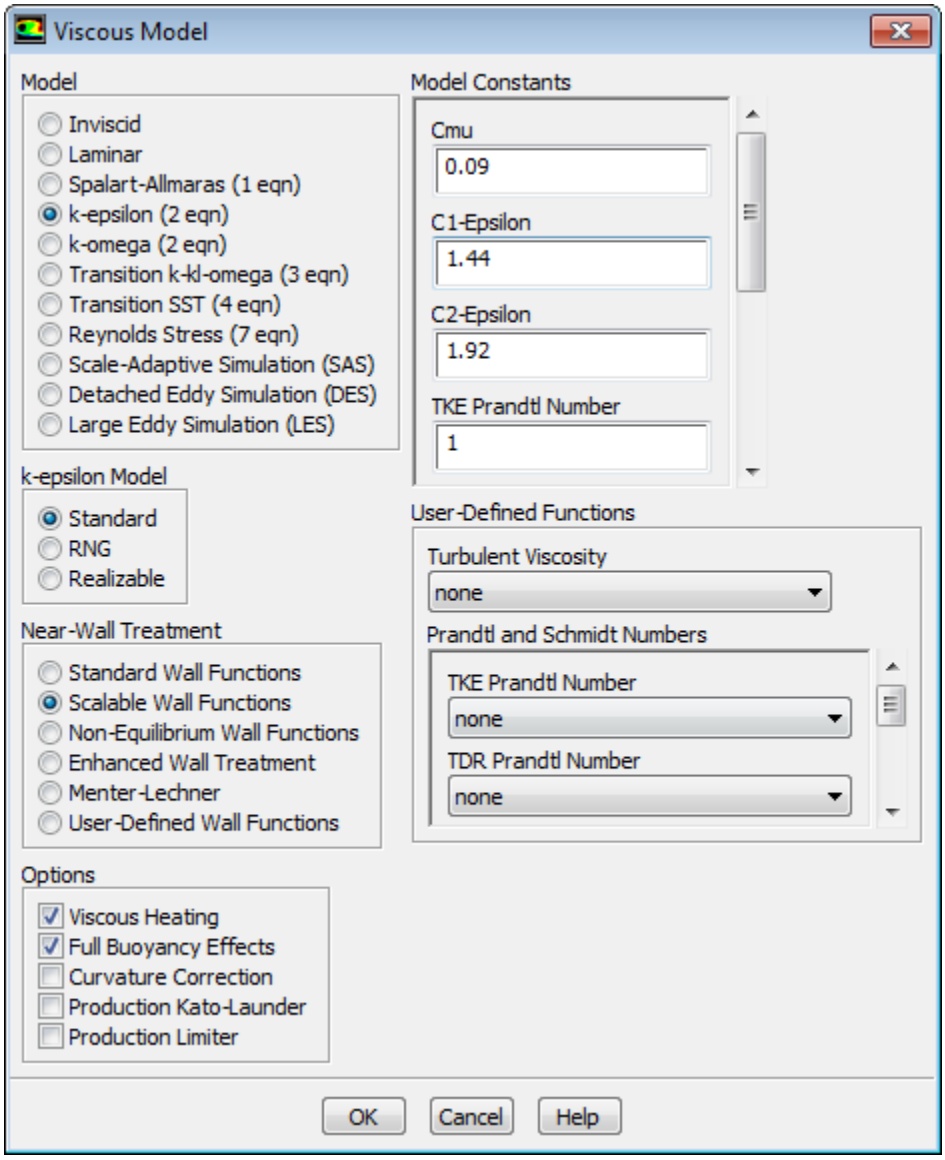
If you choose the standard k-ε model or the realizable k-ε model, the following options are available:

- viscous heating (always enabled for the density-based solvers) (Including the Viscous Heating Effects (p. 798))
- inclusion of buoyancy effects on ε (see Effects of Buoyancy on Turbulence in the k-ε Models in the Theory Guide)
- inclusion of curvature correction (Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models (p. 798))

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- inclusion of compressibility effects (Including the Compressibility Effects Option (p. 799))
- inclusion of production limiters (Including Production Limiters for Two-Equation Models (p. 799))

Figure 11.4: The Viscous Model Dialog Box Displaying the Standard k-ε Model



For all k-ε models, one of the following near-wall treatments must be selected (see Near-Wall Treatments for Wall-Bounded Turbulent Flows in the Theory Guide):

- standard wall functions
- scalable wall functions
- non-equilibrium wall functions
- enhanced wall treatment
- Menter-Lechner
- user-defined wall functions

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you choose the enhanced wall treatment, the following options are available:

- pressure gradient effects (Including Pressure Gradient Effects (p. 802))
- thermal effects (Including Thermal Effects (p. 802))

If you select user-defined wall functions for the near-wall treatment, hook your UDF using the **Law of the Wall** drop-down list that appears at the bottom of the **User-Defined Functions** group box in the **Viscous Model** dialog box.

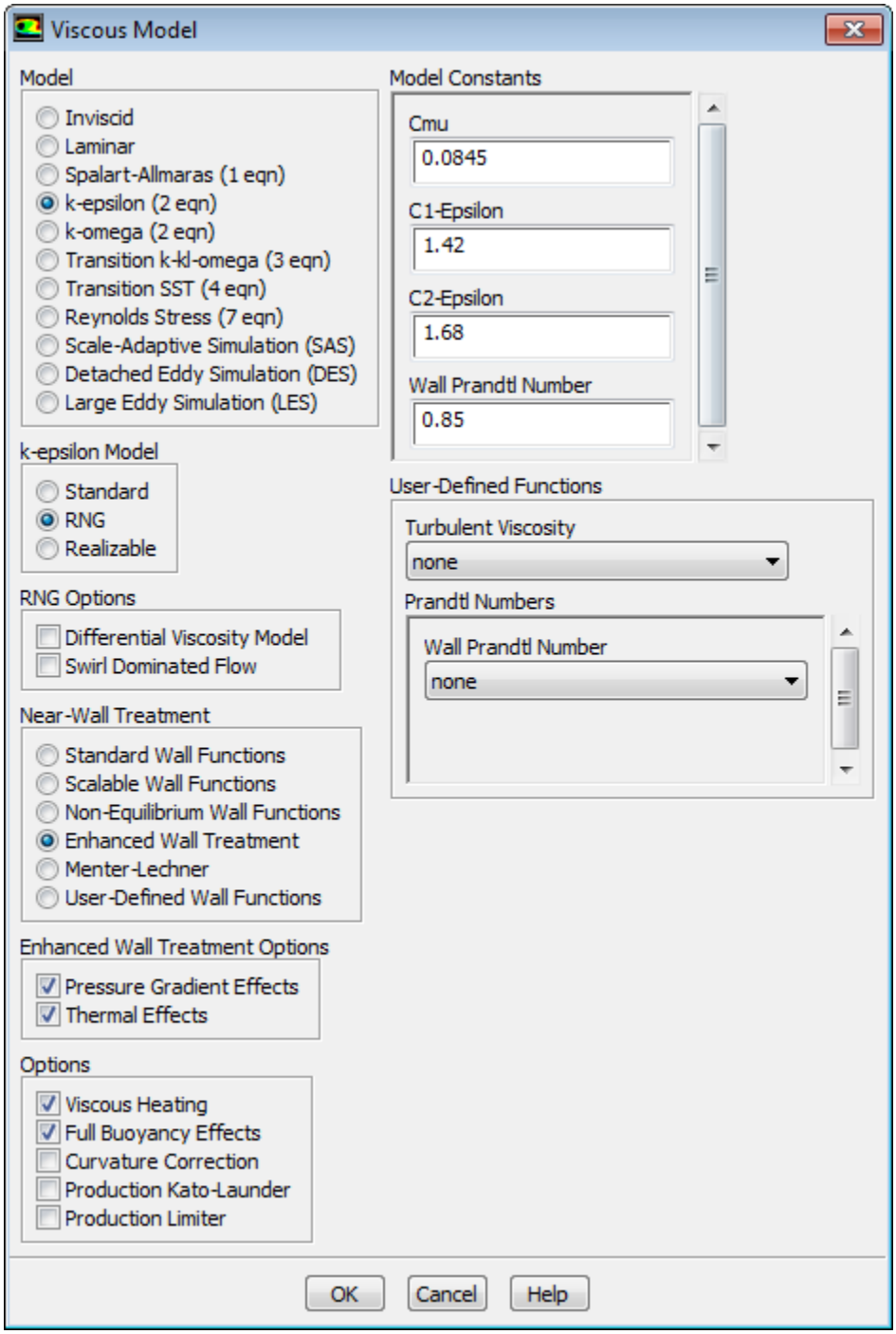
11.5.2. Setting Up the RNG k-ε Model

If you choose the RNG k-ε model, the following options are available:

- differential viscosity model (Differential Viscosity Modification (p. 800))
- swirl modification (Swirl Modification (p. 801))
- viscous heating (always enabled for the density-based solvers) (Including the Viscous Heating Effects (p. 798))
- inclusion of buoyancy effects on ε (see Effects of Buoyancy on Turbulence in the k-ε Models in the Theory Guide)
- inclusion of curvature correction (Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models (p. 798))
- inclusion of compressibility effects (Including the Compressibility Effects Option (p. 799))
- inclusion of production limiters (Including Production Limiters for Two-Equation Models (p. 799))

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.5: The Viscous Model Dialog Box Displaying the RNG k-ε Model



For all k-ε models, one of the following near-wall treatments must be selected (see Near-Wall Treatments for Wall-Bounded Turbulent Flows in the Theory Guide):

- standard wall functions
- scalable wall functions
- non-equilibrium wall functions
- enhanced wall treatment

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Menter-Lechner
- user-defined wall functions

If you choose the enhanced wall treatment, the following options are available:

- pressure gradient effects (Including Pressure Gradient Effects (p. 802))
- thermal effects (Including Thermal Effects (p. 802))

If you select user-defined wall functions for the near-wall treatment, hook your UDF using the **Law of the Wall** drop-down list that appears at the bottom of the **User-Defined Functions** group box in the **Viscous Model** dialog box.

11.6. Setting Up the k-ω Model

For additional information, see the following sections:
11.6.1. Setting Up the Standard k-ω Model
11.6.2. Setting Up the Baseline (BSL) k-ω Model
11.6.3. Setting Up the Shear-Stress Transport k-ω Model

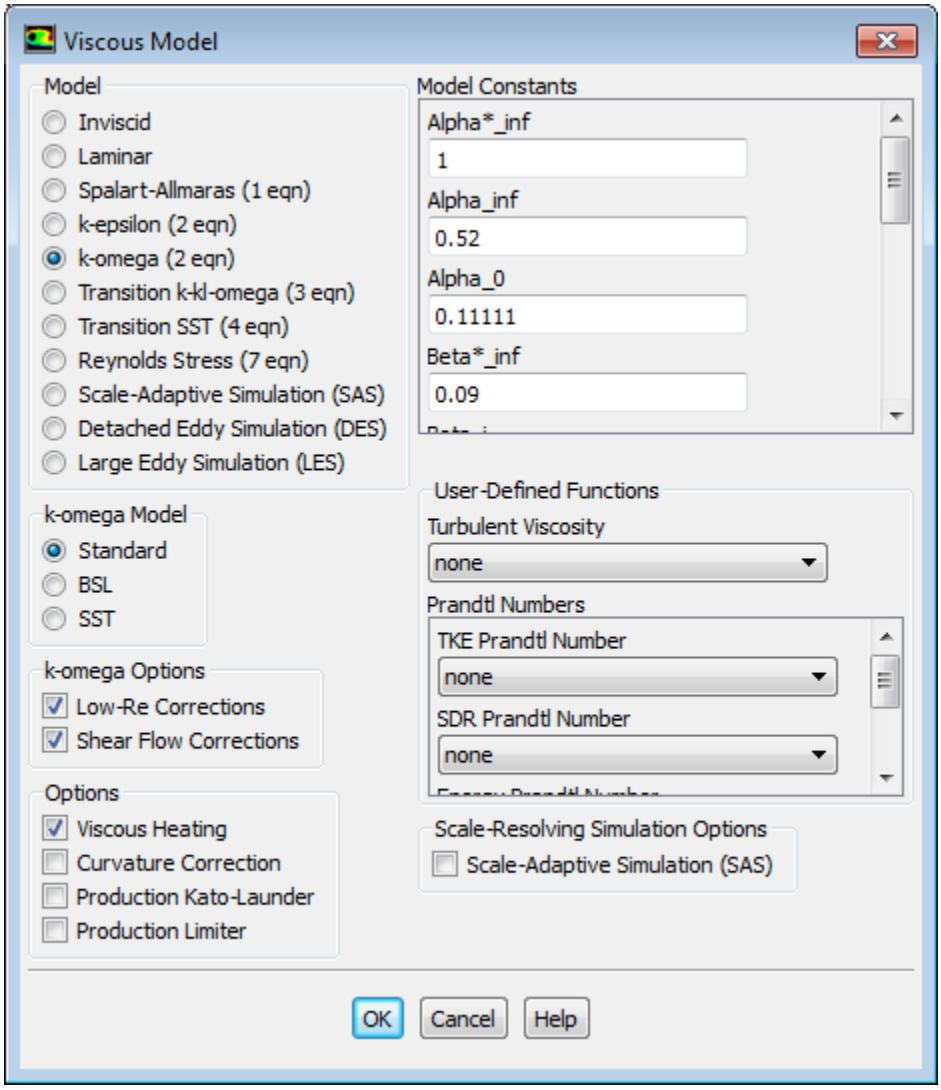
11.6.1. Setting Up the Standard k-ω Model

If you choose the standard k-ω model, the following options are available:

- low-Re corrections (Low-Re Corrections (p. 801))
- shear flow corrections (Shear Flow Corrections (p. 801))
- turbulence damping (available with the VOF and Mixture models and the Eulerian multiphase model when using the immiscible fluid model) (Turbulence Damping (p. 801))
- viscous heating (always enabled for the density-based solvers) (Including the Viscous Heating Effects (p. 798))
- inclusion of curvature correction (Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models (p. 798))
- inclusion of compressibility effects (Including the Compressibility Effects Option (p. 799))
- inclusion of production limiters (Including Production Limiters for Two-Equation Models (p. 799))
- a scale-resolving simulation option: Scale-Adaptive Simulation (Setting Up Scale-Adaptive Simulation (SAS) Modeling (p. 781))

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.6: The Viscous Model Dialog Box Displaying the Standard k-ω Model



The k-ω models use y^+-insensitive near-wall treatment (see y+-Insensitive Wall Treatment ω-Equation in the Theory Guide).

11.6.2. Setting Up the Baseline (BSL) k-ω Model

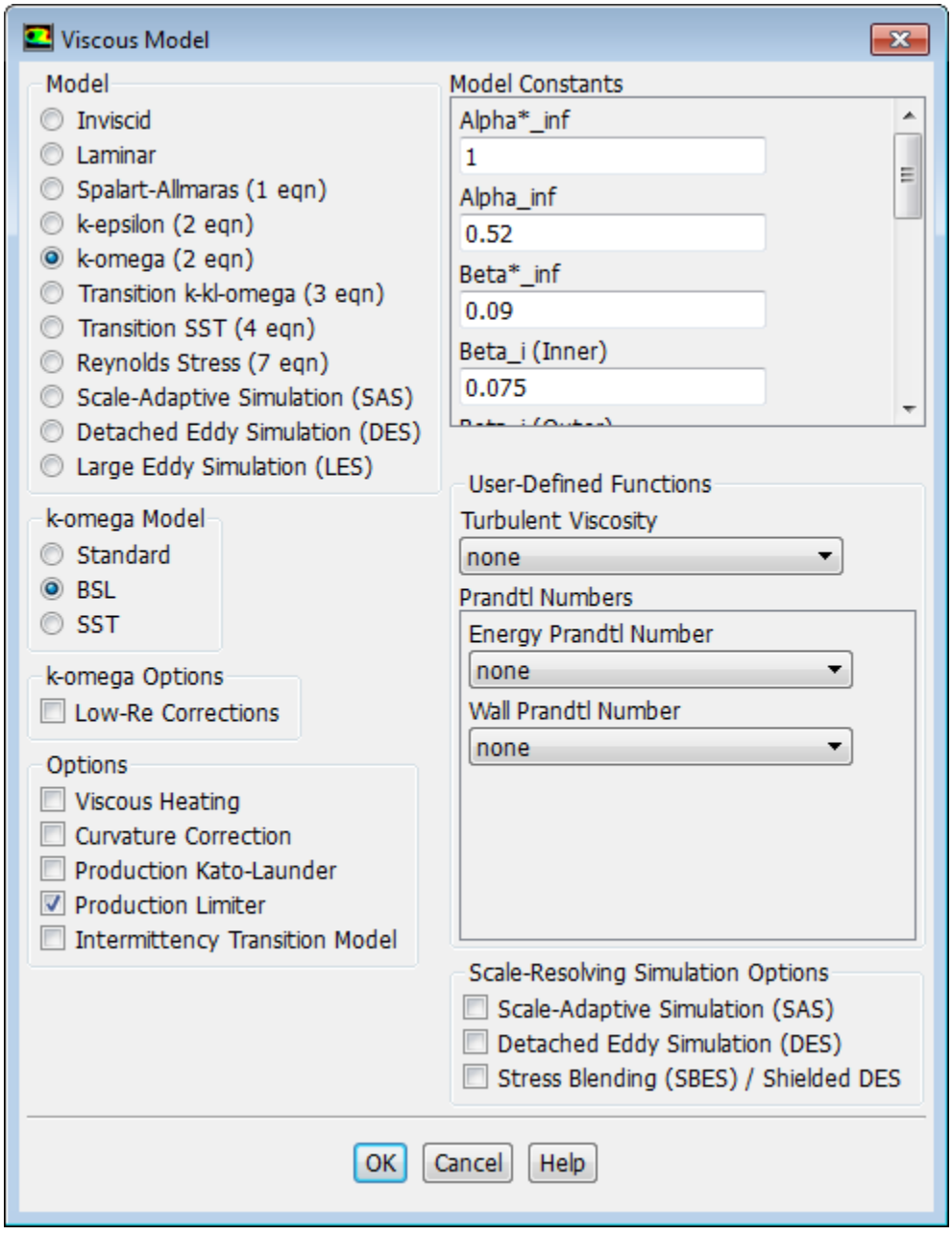
If you choose the baseline (BSL) k-ω model, the following options are available:

- low-Re corrections (Low-Re Corrections (p. 801))
- turbulence damping (available with the VOF and Mixture models and the Eulerian multiphase model when using the immiscible fluid model) (Turbulence Damping (p. 801))
- viscous heating (always enabled for the density-based solvers) (Including the Viscous Heating Effects (p. 798))
- inclusion of curvature correction (Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models (p. 798))
- inclusion of compressibility effects (Including the Compressibility Effects Option (p. 799))
- inclusion of production limiters (Including Production Limiters for Two-Equation Models (p. 799))

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- inclusion of the Intermittency Transition model
- scale-resolving simulation options: either Scale-Adaptive Simulation (Setting Up Scale-Adaptive Simulation (SAS) Modeling (p. 781)), Detached Eddy Simulation (Setting Up DES with the Transition SST Model (p. 789)), Stress-Blended Eddy Simulation (Including the SDES or SBES Model with BSL, SST, and Transition SST Models (p. 805)), or Shielded Detached Eddy Simulation (Including the SDES or SBES Model with BSL, SST, and Transition SST Models (p. 805)).

Figure 11.7: The Viscous Model Dialog Box Displaying the BSL k-ω Model



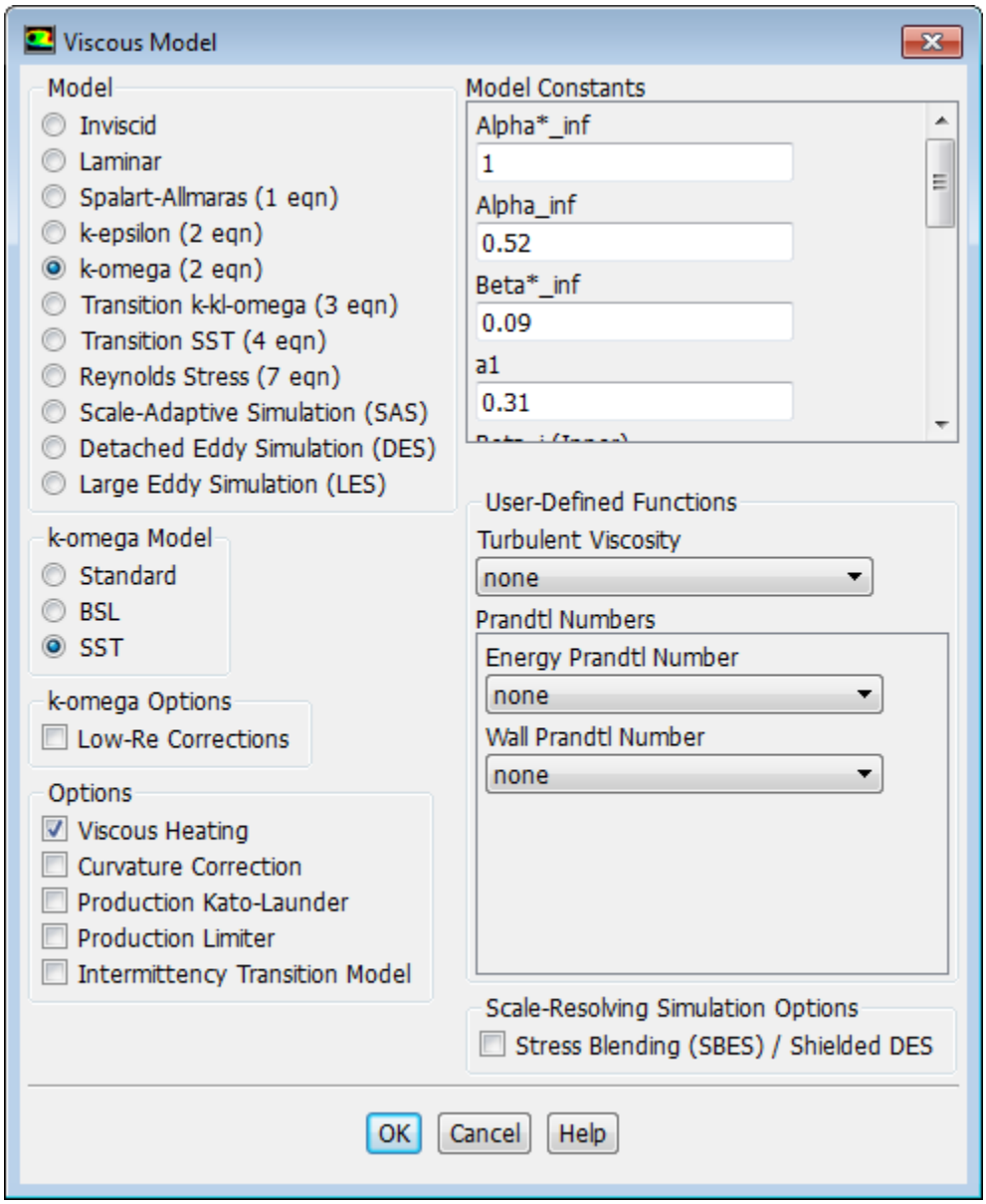
11.6.3. Setting Up the Shear-Stress Transport k-ω Model

If you choose the shear-stress transport k-ω model, the following options are available:

- low-Re corrections (Low-Re Corrections (p. 801))

- turbulence damping (available with the VOF and Mixture models and the Eulerian multiphase model when using the immiscible fluid model) (Turbulence Damping (p. 801))
- viscous heating (always enabled for the density-based solvers) (Including the Viscous Heating Effects (p. 798))
- inclusion of curvature correction (Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models (p. 798))
- inclusion of compressibility effects (Including the Compressibility Effects Option (p. 799))
- inclusion of production limiters (Including Production Limiters for Two-Equation Models (p. 799))
- inclusion of the Intermittency Transition model
- scale-resolving simulation options: Stress-Blended Eddy Simulation or Shielded Detached Eddy Simulation (Including the SDES or SBES Model with BSL, SST, and Transition SST Models (p. 805)).

Figure 11.8: The Viscous Model Dialog Box Displaying the SST k-ω Model



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.7. Setting Up the Transition k-kl-ω Model

If you choose the Transition k-kl-ω model, it is not necessary to modify any of the model constants.

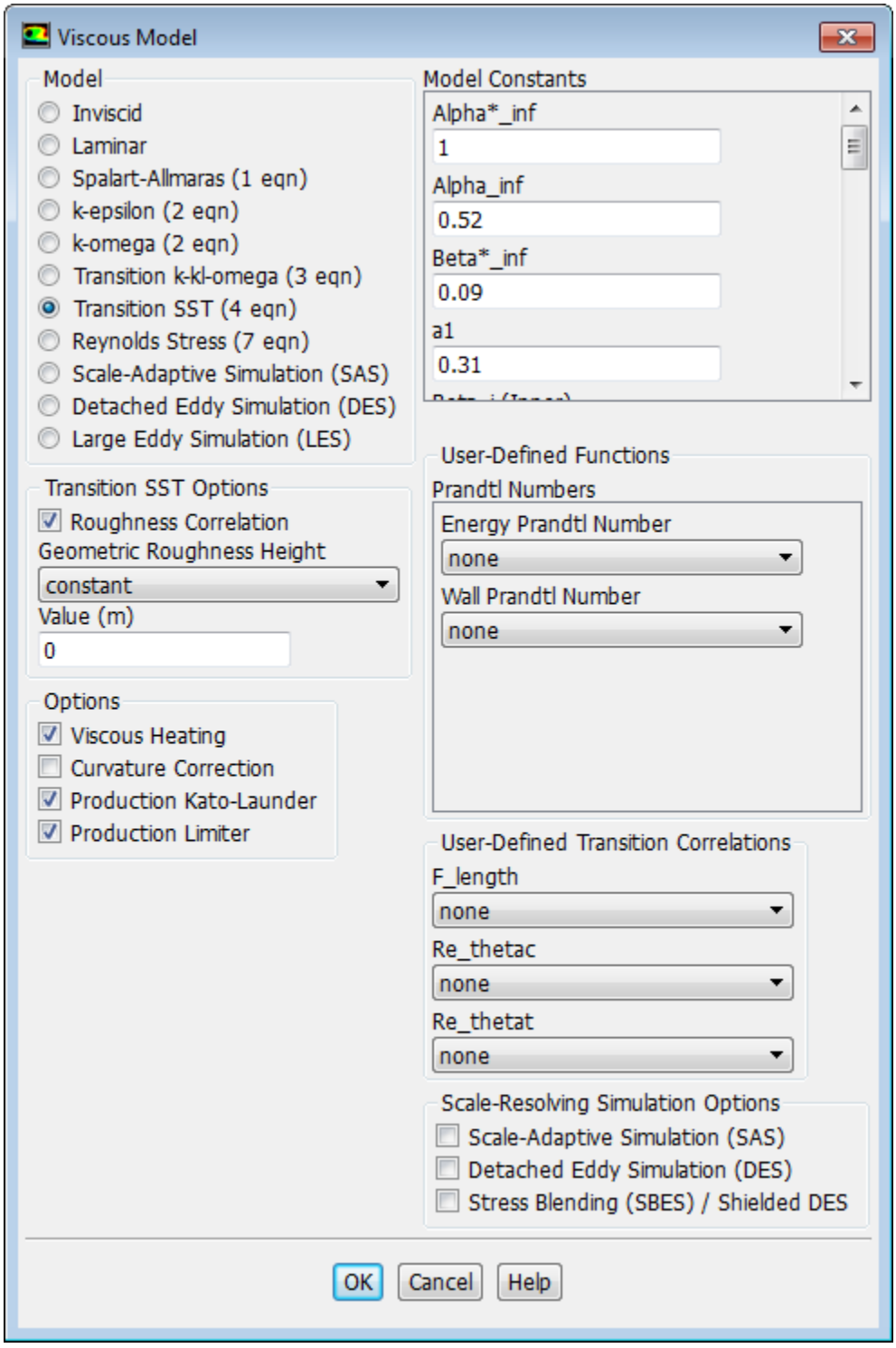
11.8. Setting Up the Transition SST Model

If you choose the Transition SST model (also known as the γ-Re_θ model), the following options are available:

- roughness correlation (Transition SST and Rough Walls in the Theory Guide)
- viscous heating (always enabled for the density-based solvers) (Including the Viscous Heating Effects (p. 798))
- inclusion of curvature correction (Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models (p. 798))
- inclusion of production limiters (Including Production Limiters for Two-Equation Models (p. 799))
- scale-resolving simulation options: either Scale-Adaptive Simulation (Setting Up Scale-Adaptive Simulation (SAS) Modeling (p. 781)), Detached Eddy Simulation (Setting Up DES with the Transition SST Model (p. 789)), Stress-Blended Eddy Simulation (Including the SDES or SBES Model with BSL, SST, and Transition SST Models (p. 805)), or Shielded Detached Eddy Simulation (Including the SDES or SBES Model with BSL, SST, and Transition SST Models (p. 805)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.9: The Viscous Model Dialog Box for the Transition SST Model



You can customize your transition correlations, which are used in conjunction with the Transition SST model. The user-defined functions that you can hook are:

- transition length function (**F_length**).
- critical momentum thickness Reynolds number (**Re_thetac**).
- transition onset momentum thickness Reynolds number (**Re_thetat**).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For detailed information about the transition correlation UDFs, see DEFINE_TRANS UDFs in the Fluent Customization Manual.

11.9. Setting Up the Intermittency Transition Model

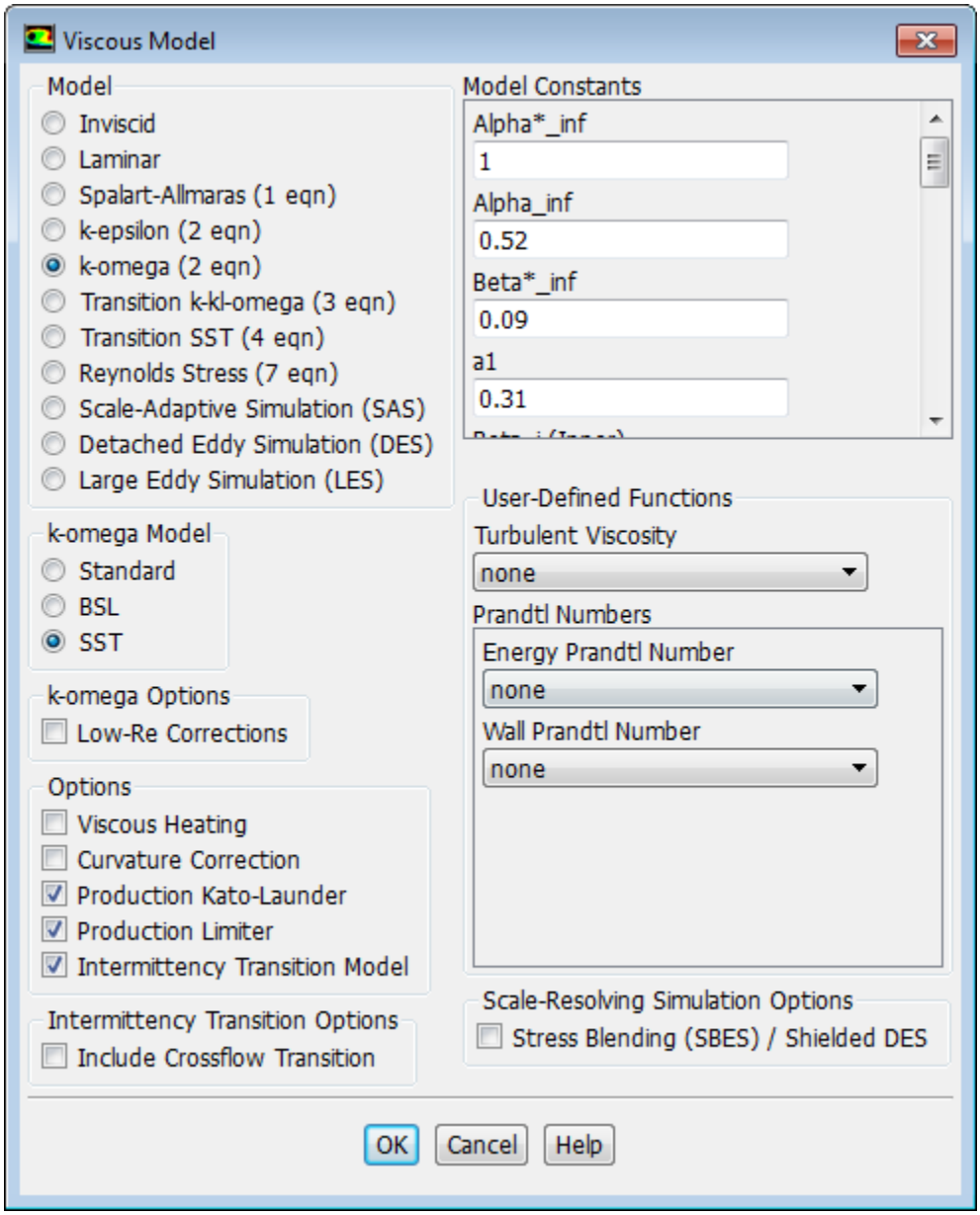
The Intermittency Transition model (also known as the γ model) is available as an option for the following turbulence models:

- BSL k-ω model
- SST k-ω model
- Scale-Adaptive Simulation with BSL / SST
- Detached Eddy Simulation with BSL / SST
- Shielded Detached Eddy Simulation (SDES) with BSL or SST
- Stress-Blended Eddy Simulation (SBES) with BSL / SST

To apply it in combination with one of these turbulence models, you must enable the **Intermittency Transition Model** option from the **Options** group box in the **Viscous Model** dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.10: The Intermittency Transition Model in Combination with the SST k-ω Model



After you have enabled the Intermittency Transition model, the additional option **Include Crossflow Transition** will be available. This additional option allows you to include the effects of crossflow instability.

11.10. Setting Up the Reynolds Stress Model

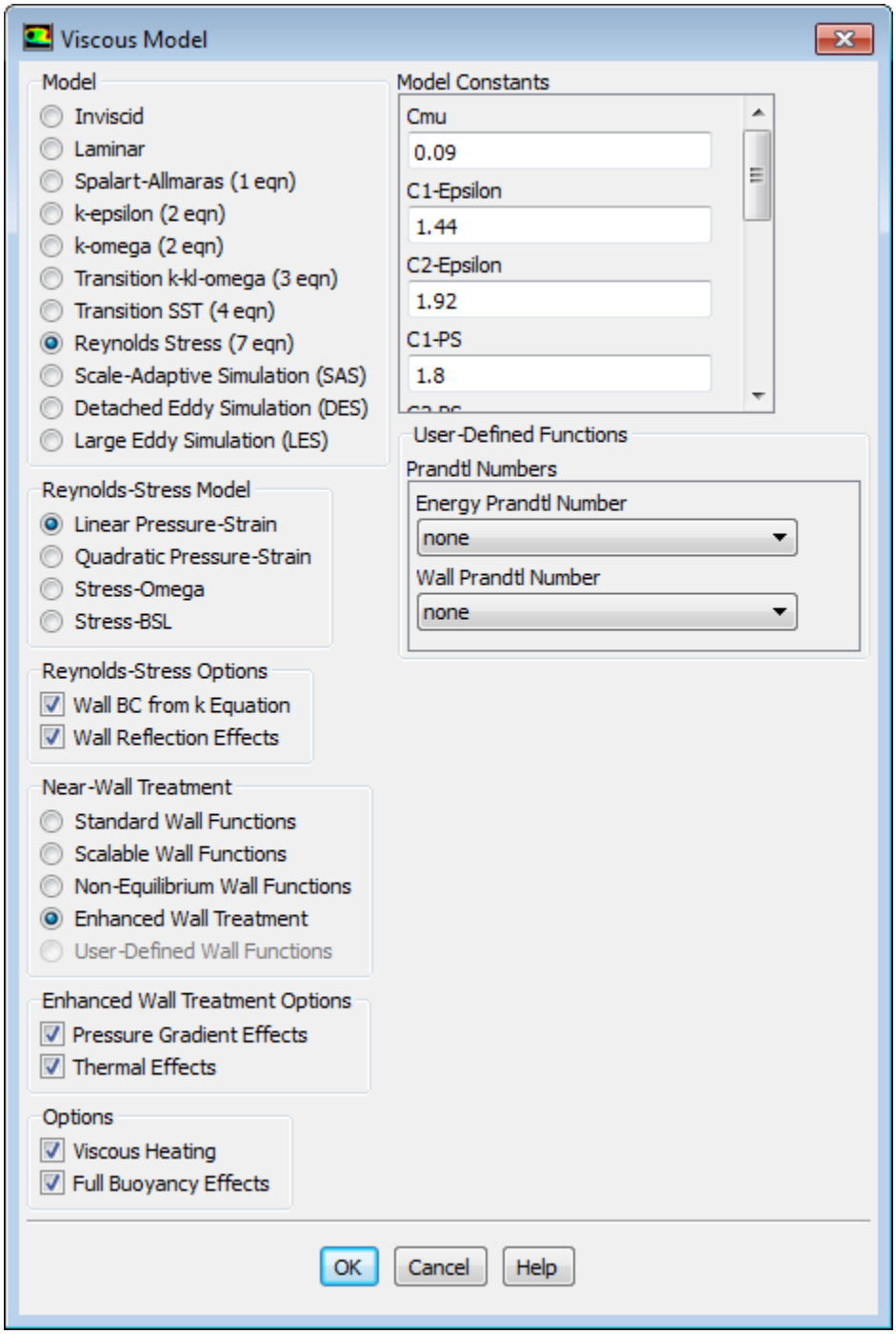
If you choose the RSM, the following submodels are available:

- ε-based Reynolds stress models:
  - linear pressure-strain model (see Linear Pressure-Strain Model in the Theory Guide)
  - quadratic pressure-strain model (see Quadratic Pressure-Strain Model in the Theory Guide)
- ω-based Reynolds stress models:
  - Stress-Omega (see Stress-Omega Model in the Theory Guide)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– Stress-BSL (see Stress-BSL Model in the Theory Guide)

Figure 11.11: The Viscous Model Dialog Box Displaying the Reynolds Stress Model Options



The following options are specific to the Reynolds-stress models based on the ε-equation:

- wall boundary conditions for the Reynolds stresses from the k equation (Solving the k Equation to Obtain Wall Boundary Conditions (p. 802)) for the linear and quadratic pressure-strain models
- wall reflection effects on Reynolds stresses (Including the Wall Reflection Term (p. 802)) for the linear pressure-strain model
- near-wall treatments (see Near-Wall Treatments for Wall-Bounded Turbulent Flows in the Theory Guide):

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - standard wall functions
  - scalable wall functions
  - non-equilibrium wall functions
  - enhanced wall treatment (only for the linear pressure-strain model)

If you choose the enhanced wall treatment, the following options are available:

- pressure gradient effects (Including Pressure Gradient Effects (p. 802))
- thermal effects (Including Thermal Effects (p. 802))

For the Reynolds stress models based on the ω- (**Stress-Omega**) and BSL-equation (**Stress-BSL**), a near-wall treatment is automatically used to perform a blending between the viscous sublayer and the logarithmic region, and therefore near-wall treatment options are not shown. A low Re-number (LRN) mesh is recommended for these models.

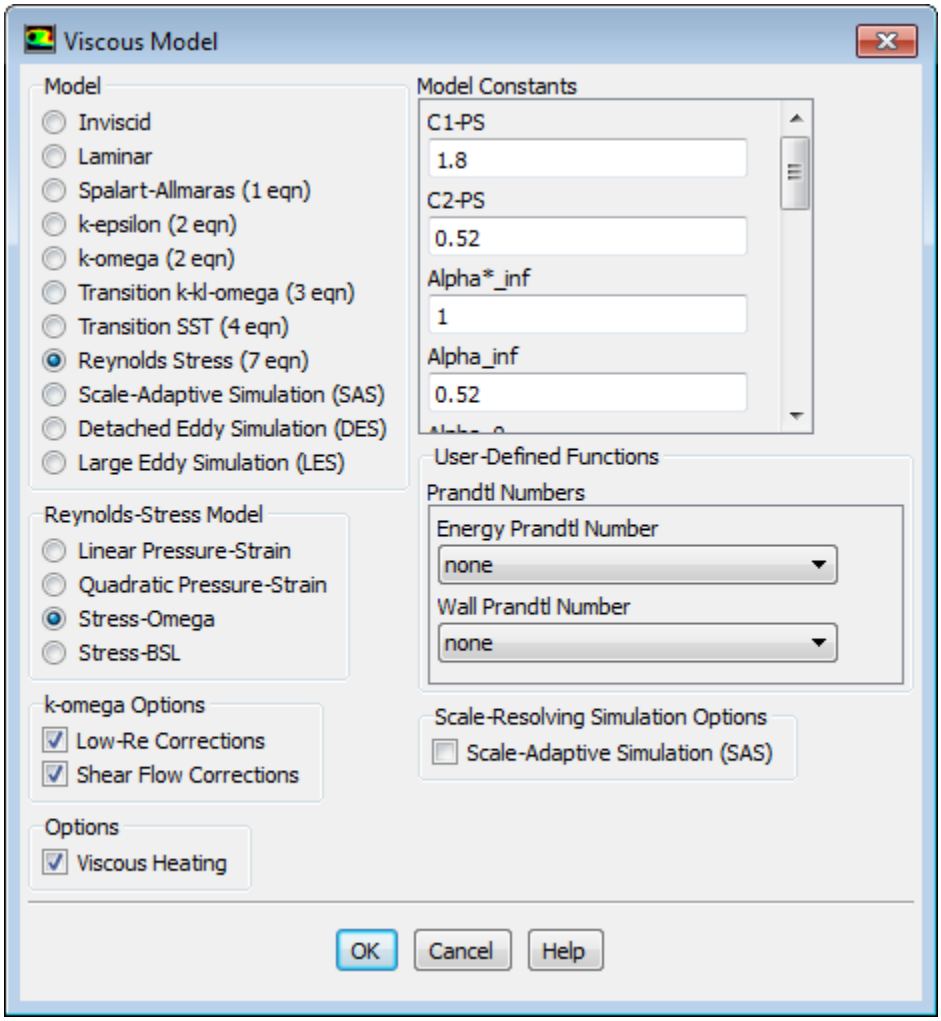
For **Stress-Omega**, you do have the option of selecting any or all of the following $k-\omega$ options:

- low-Re corrections (Low-Re Corrections (p. 801))
- shear flow corrections (Shear Flow Corrections (p. 801))

For both **Stress-Omega** and **Stress-BSL**, the following scale-resolving simulation option is available: Scale-Adaptive Simulation (Setting Up Scale-Adaptive Simulation (SAS) Modeling (p. 781)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.12: The Viscous Model Dialog Box Displaying the Stress-Omega Model Options



Other options that are available for both ε- and ω-based Reynolds stress models depending on your case setup include:

- viscous heating (always enabled for the density-based solvers) (Including the Viscous Heating Effects (p. 798))
- inclusion of compressibility effects (Including the Compressibility Effects Option (p. 799))
- inclusion of buoyancy effects on ε (see Effects of Buoyancy on Turbulence in the k-ε Models in the Theory Guide)

Note

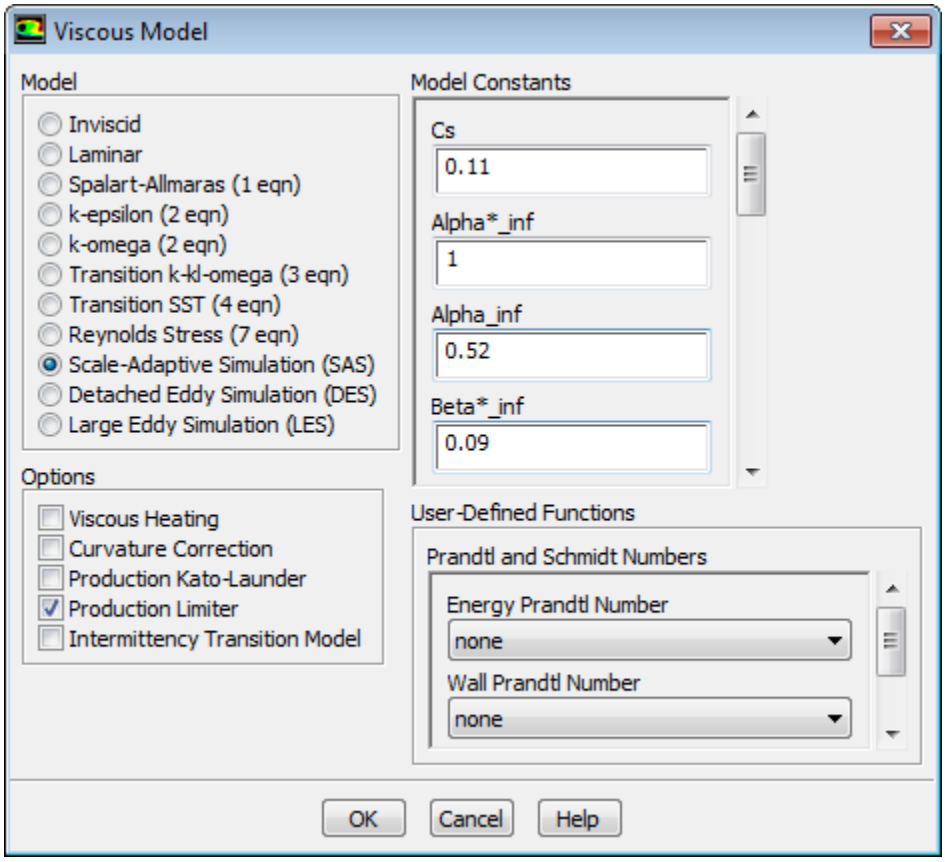
The ω-based Reynolds stress models are not compatible with the Eulerian multiphase model.

11.11. Setting Up Scale-Adaptive Simulation (SAS) Modeling

Scale-Adaptive Simulation (SAS) modeling is an approach for the simulation of unsteady turbulent flows, and can be applied in combination with most ω-based URANS turbulence models. To apply it in combination with the SST k-ω turbulence model, you can simply select **Scale-Adaptive Simulation (SAS)** from the **Model** list in the **Viscous Model** dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

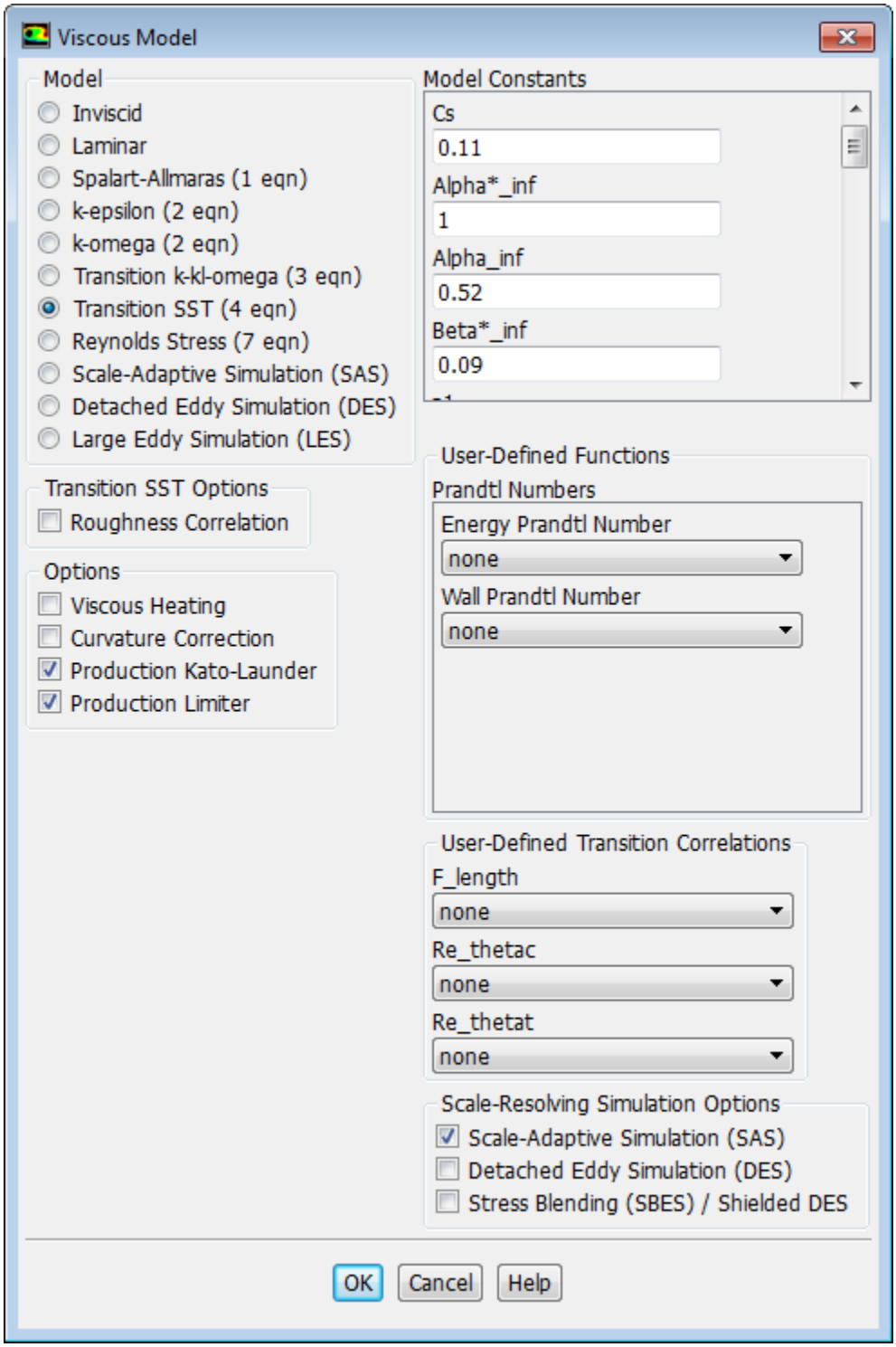
Figure 11.13: Scale-Adaptive Simulation (SAS) in Combination with the SST Turbulence Model



You can also apply SAS in combination with the following ω-based URANS models: the Standard k-ω model, the BSL k-ω model, the Transition SST model, and the ω-based Reynolds stress models (RSM). Simply select the appropriate **Model** and then enable the **Scale-Adaptive Simulation (SAS)** option in the **Scale-Resolving Simulation Options** group box. For an example, see Figure 11.14: Scale-Adaptive Simulation (SAS) in Combination with the Transition SST Model (p. 783).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.14: Scale-Adaptive Simulation (SAS) in Combination with the Transition SST Model



Note that for SAS modeling, the **Bounded Central Differencing** scheme (available in the **Solution Methods** task page) is recommended for momentum discretization, and is the default setting.

11.12. Setting Up the Detached Eddy Simulation Model

When using the Detached Eddy Simulation (DES) model, the following underlying RANS turbulence models can be used:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Spalart-Allmaras
- Realizable k-ε
- SST k-ω
- BSL k-ω
- Transition SST

Note that for the DES model, the **Bounded Central Differencing** scheme (available in the **Solution Methods** task page) is recommended for momentum discretization, and is the default setting.

For additional information, see the following sections:
11.12.1. Setting Up DES with the Spalart-Allmaras Model
11.12.2. Setting Up DES with the Realizable k-ε Model
11.12.3. Setting Up DES with the SST k-ω Model
11.12.4. Setting Up DES with the BSL k-ω Model
11.12.5. Setting Up DES with the Transition SST Model

11.12.1. Setting Up DES with the Spalart-Allmaras Model

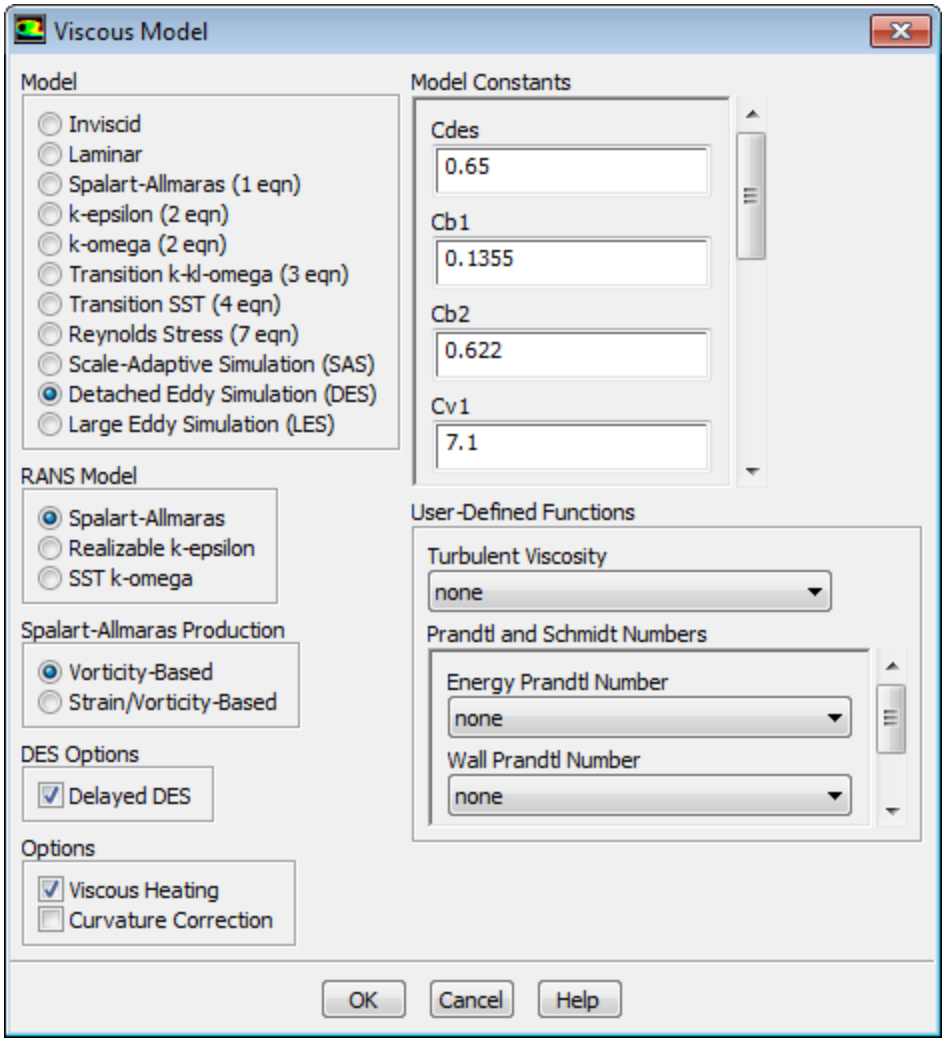
To set up a Detached Eddy Simulation with the Spalart-Allmaras model, select **Detached Eddy Simulation (DES)** from the **Model** list in the **Viscous Model** dialog box and then select **Spalart-Allmaras** from the **RANS Model** list.

ANSYS Fluent uses Equation 4.249 (in the Theory Guide) to compute the value of the length scale $\tilde{d}$ for the Spalart-Allmaras model. By default, the empirical constant C_{des} is set to 0.65. You can change its value in the **Cdes** field under **Model Constants**. The following options are available for this model:

- vorticity-based production (Vorticity- and Strain/Vorticity-Based Production (p. 800))
- strain/vorticity-based production (Vorticity- and Strain/Vorticity-Based Production (p. 800))
- delayed DES (Delayed Detached Eddy Simulation (DDES) (p. 800))
- viscous heating (always enabled for the density-based solvers) (Including the Viscous Heating Effects (p. 798))
- inclusion of curvature correction (Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models (p. 798))

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.15: The Viscous Model Dialog Box Displaying Options for DES with the Spalart-Allmaras Model



Additionally, you can perform the following DES-specific function by using the `/define/models/viscous/detached-eddy-simulation?` text command:

- Modify only the length scales that appear in the destruction term in ν_t equation (the default is to modify all length scales within the ν_t equation)

11.12.2. Setting Up DES with the Realizable k-ε Model

To set up a Detached Eddy Simulation with the Realizable k-ε model, select **Detached Eddy Simulation (DES)** from the **Model** list in the **Viscous Model** dialog box and then select **Realizable k-epsilon** from the **RANS Model** list.

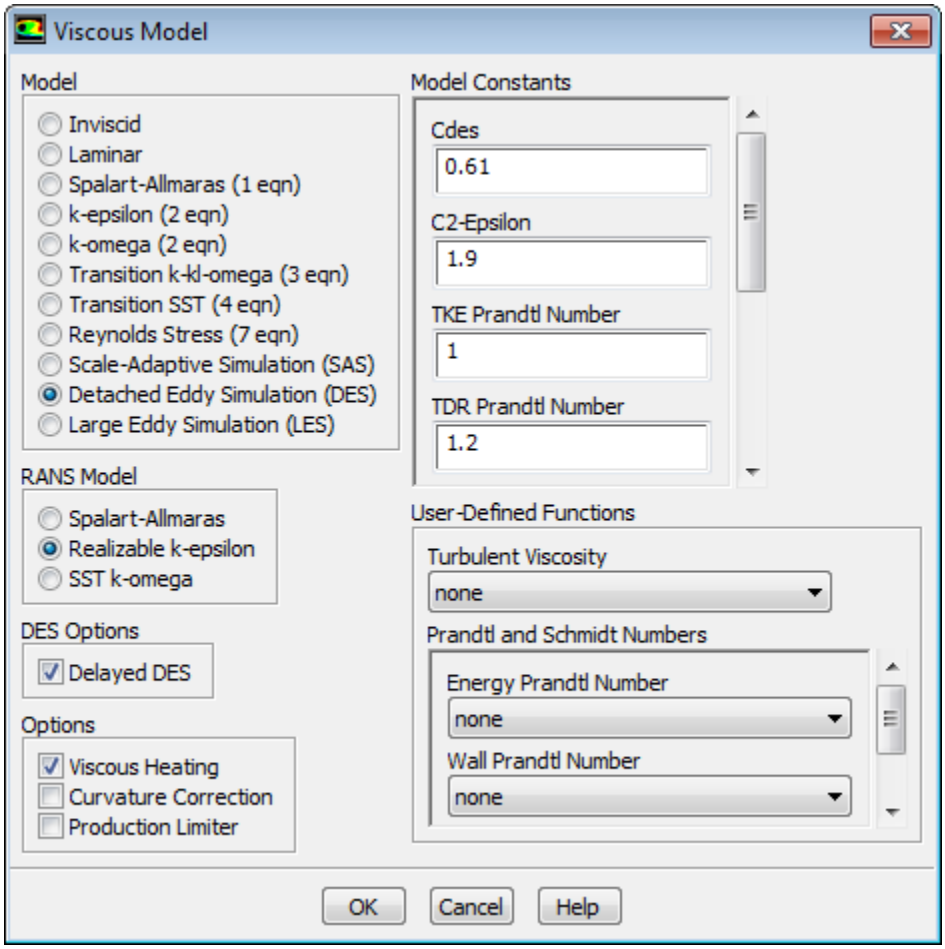
The model-specific options for DES with the Realizable k-ε model are the following:

- delayed DES (Delayed Detached Eddy Simulation (DDES) (p. 800))
- viscous heating (always enabled for the density-based solvers) (Including the Viscous Heating Effects (p. 798))
- inclusion of compressibility effects (Including the Compressibility Effects Option (p. 799))

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The model constant C_{des} is set to 0.61 for the Realizable k-ε model (see DES with the Realizable k-ε Model in the Theory Guide). The enhanced wall treatment (EWT-ε) is always enabled for DES with the Realizable k-ε model (see Enhanced Wall Treatment ε-Equation (EWT-ε) in the Theory Guide).

Figure 11.16: The Viscous Model Dialog Box Displaying Options for DES with the Realizable k-ε Model



11.12.3. Setting Up DES with the SST k-ω Model

To set up a Detached Eddy Simulation with the SST k-ω model, select **Detached Eddy Simulation (DES)** from the **Model** list in the **Viscous Model** dialog box and then select **SST k-omega** from the **RANS Model** list.

The model-specific options that you can select for DES with the SST k-ω model are the following:

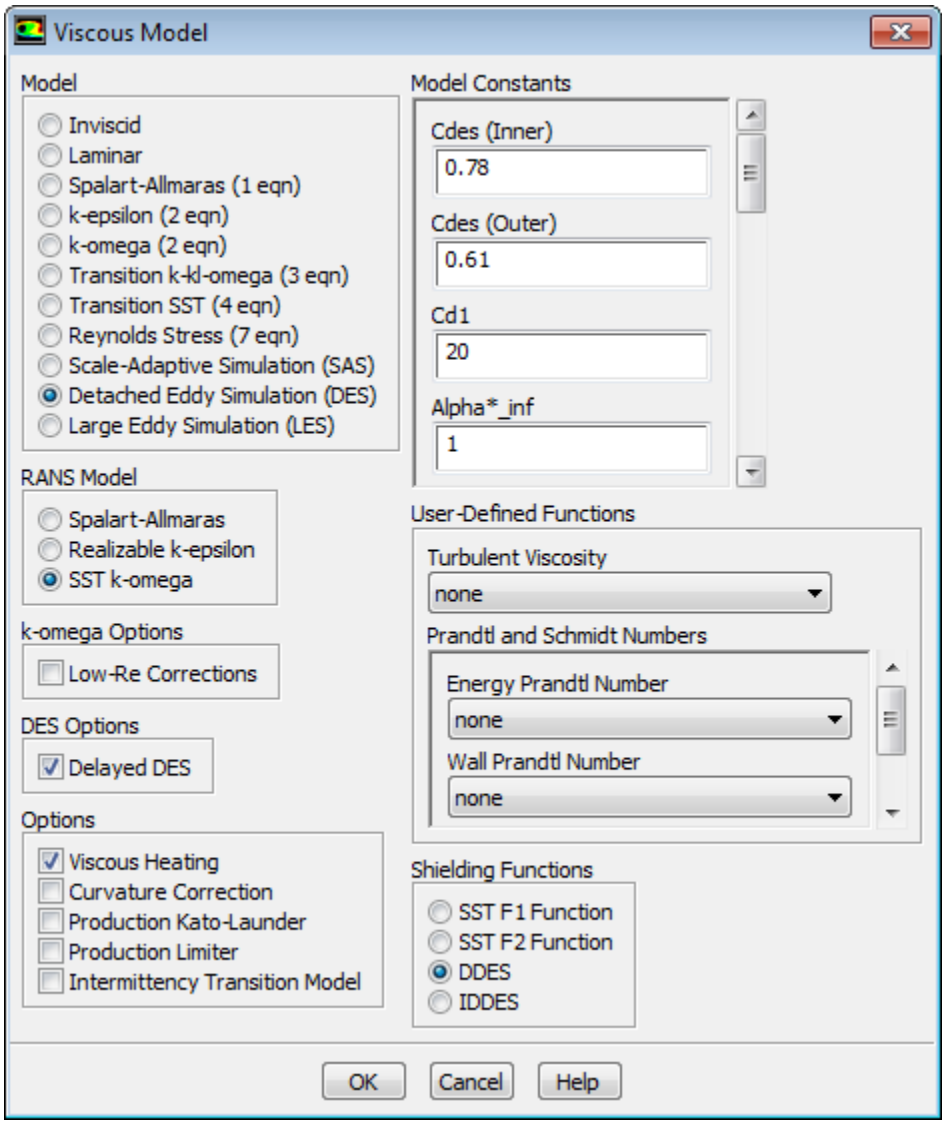
- low-Re corrections k-ω option (Low-Re Corrections (p. 801))
- delayed DES (Delayed Detached Eddy Simulation (DDES) (p. 800))
- viscous heating (always enabled for the density-based solvers) (Including the Viscous Heating Effects (p. 798))
- inclusion of curvature correction (Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models (p. 798))
- inclusion of production limiters (Including Production Limiters for Two-Equation Models (p. 799))

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- inclusion of the Intermittency Transition model
- Shielding functions—F1, F2, DDES (Delayed DES) and IDDES (Improved Delayed DES)—are available when the delayed DES option is enabled (Shielding Functions for the BSL / SST / Transition SST Detached Eddy Simulation Model (p. 807))

The model constant C_{des} is set to 0.61 for the SST k-ω RANS model (see DES with the BSL or SST k-ω Model in the Theory Guide).

Figure 11.17: The Viscous Model Dialog Box Displaying Options for DES with the SST k-ω Model



11.12.4. Setting Up DES with the BSL k-ω Model

To set up a Detached Eddy Simulation with the BSL k-ω model for a transient case, select **k-omega** from the **Model** list in the **Viscous Model** dialog box, select **BSL** from the **k-omega Model** list, and enable the **Detached Eddy Simulation (DES)** option in the **Scale-Resolving Simulation Options** group box.

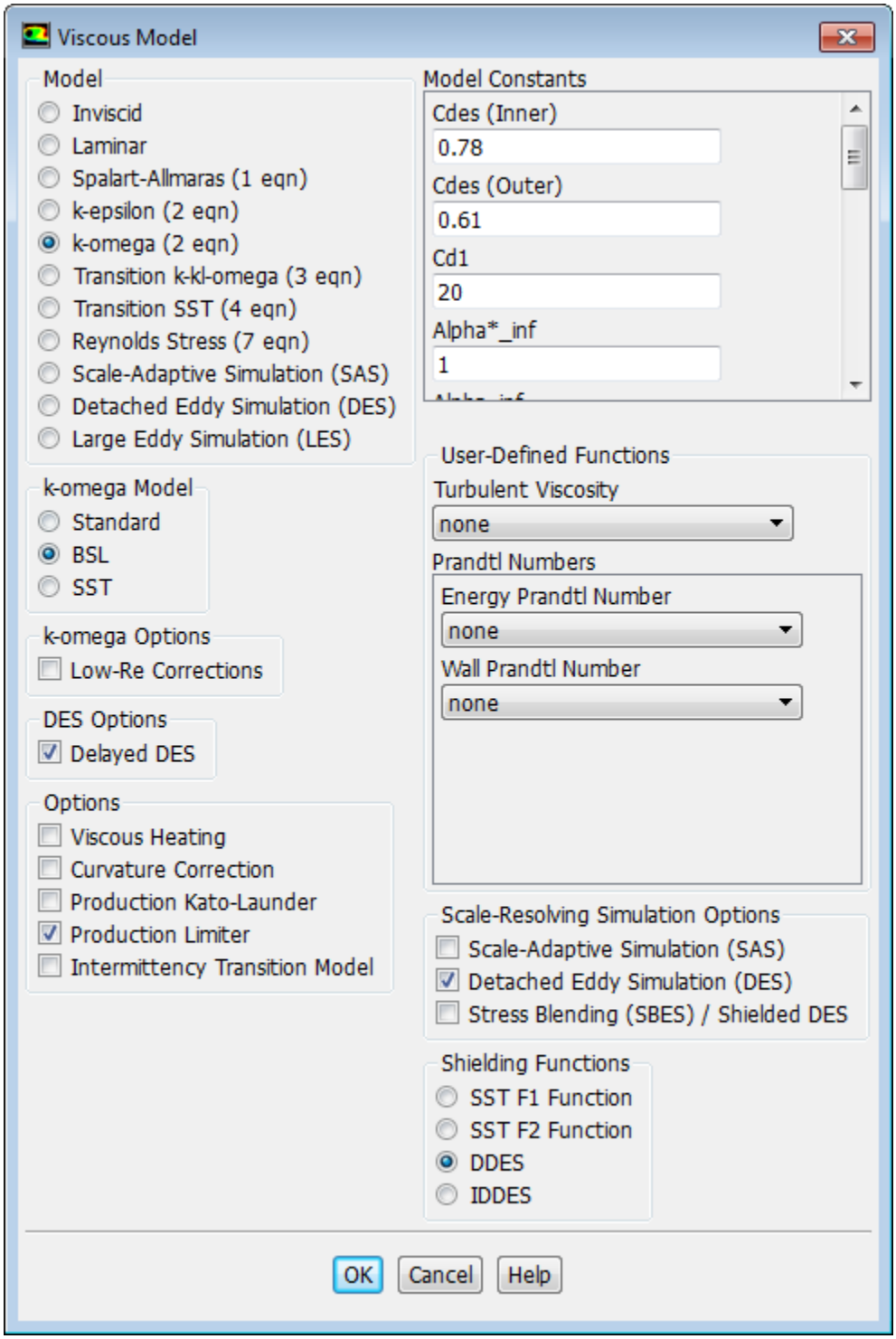
The model-specific options that you can select for DES with the BSL k-ω model are the following:

- low-Re corrections k-ω option (Low-Re Corrections (p. 801))
- delayed DES (Delayed Detached Eddy Simulation (DDES) (p. 800))
- viscous heating (always enabled for the density-based solvers) (Including the Viscous Heating Effects (p. 798))
- inclusion of curvature correction (Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models (p. 798))
- inclusion of compressibility effects (Including the Compressibility Effects Option (p. 799))
- inclusion of production limiters (Including Production Limiters for Two-Equation Models (p. 799))
- inclusion of the Intermittency Transition model
- Shielding functions—F1, F2, DDES (Delayed DES) and IDDES (Improved Delayed DES)—are available when the delayed DES option is enabled (Shielding Functions for the BSL / SST / Transition SST Detached Eddy Simulation Model (p. 807))

The model constant C_{des} is set to 0.61 for the BSL k-ω RANS model (see DES with the BSL or SST k-ω Model in the Theory Guide).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.18: The Viscous Model Dialog Box Displaying Options for DES with the BSL k-ω Model



11.12.5. Setting Up DES with the Transition SST Model

To set up a Detached Eddy Simulation with the Transition SST model for a transient case, select **Transition SST** from the **Model** list in the **Viscous Model** dialog box and then enable the **Detached Eddy Simulation (DES)** option in the **Scale-Resolving Simulation Options** group box.

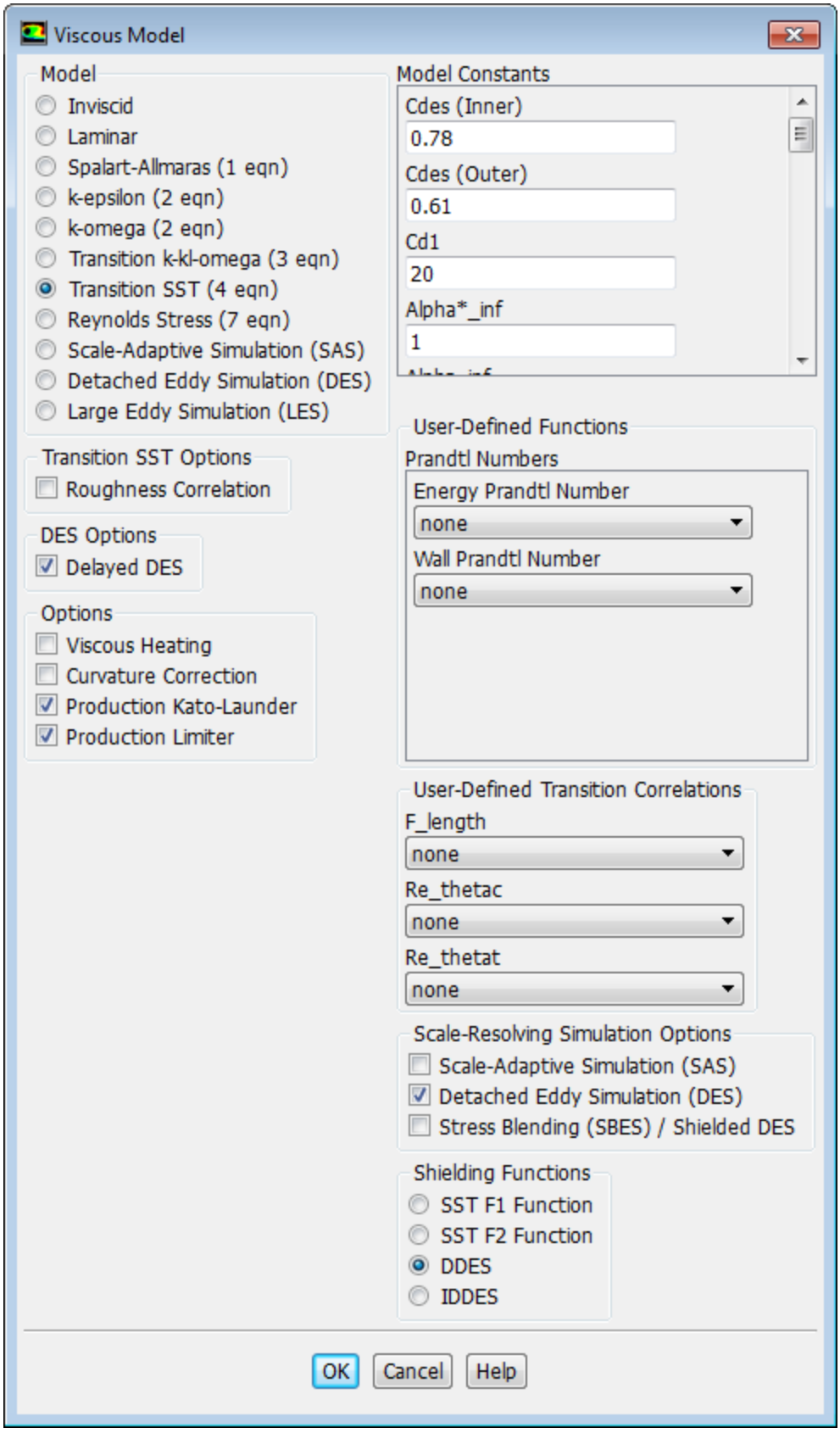
For DES with the Transition SST model, you can choose to enable the **Delayed DES** option in the **DES Options** group box (see Delayed Detached Eddy Simulation (DDES) (p. 800) for details). When this option

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

is enabled you can make a selection from the **Shielding Functions** list, as described in Shielding Functions for the BSL / SST / Transition SST Detached Eddy Simulation Model (p. 807).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.19: The Viscous Model Dialog Box Displaying Options for DES with the Transition SST Model



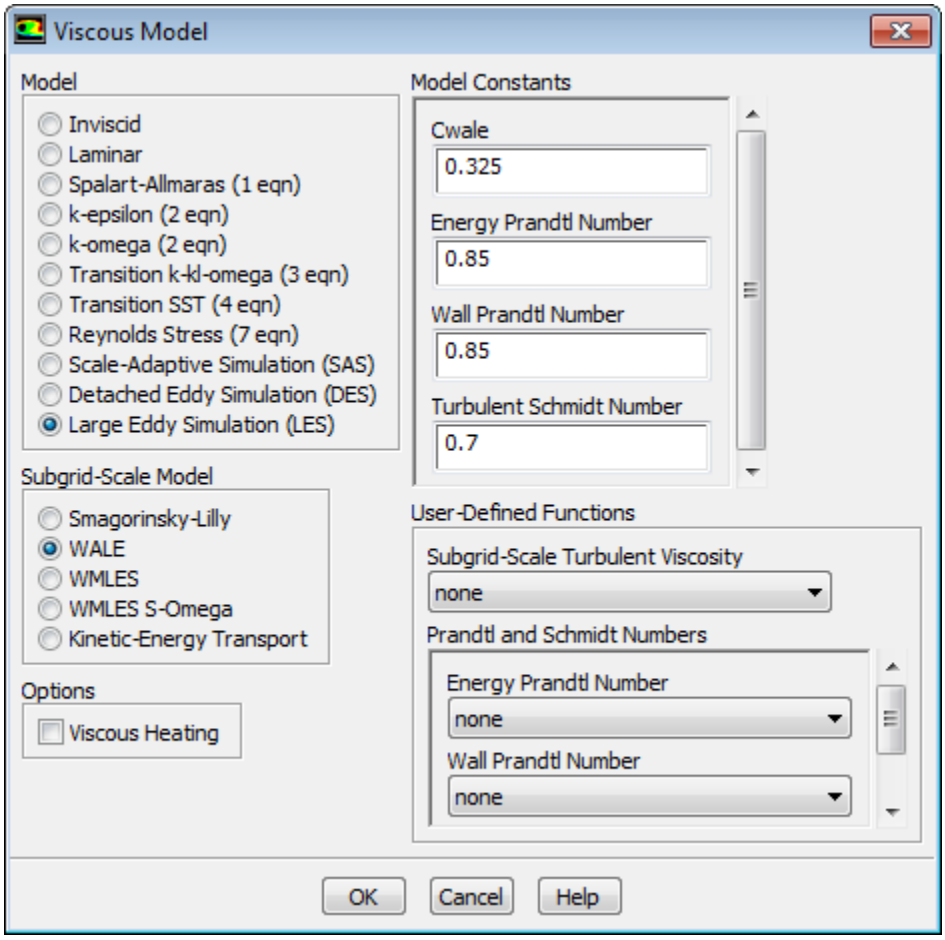
11.13. Setting Up the Large Eddy Simulation Model

If you choose the LES model, the following subgrid-scale submodels are available (Subgrid-Scale Model (p. 803)):

- **Smagorinsky-Lilly**
- **WALE**
- **WMLES**
- **WMLES S-Omega**
- **Kinetic-Energy Transport**

The default **Subgrid-Scale Model** for the LES model is the **WALE** model.

Figure 11.20: The Viscous Model Dialog Box Displaying the Large Eddy Simulation Model Options



The LES options that are available for the **Smagorinsky-Lilly** submodel are

- **Dynamic Stress**
- **Dynamic Energy Flux** (available only when the **Dynamic Stress** option is enabled)
- **Dynamic Scalar Flux** (available only when the **Dynamic Stress** option is enabled)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The LES options that are available when the **Kinetic-Energy Transport** submodel is selected are **Dynamic Energy Flux** and **Dynamic Scalar Flux**.

The **Dynamic Fvar** option is available for all of the subgrid-scale submodels when **Non-Premixed Combustion** or **Partially Premixed Combustion** is selected in the Species Model Dialog Box (p. 2216). This option enables the dynamic mixture fraction variance model. See The Non-Premixed Model for LES in the Theory Guide for details.

When using the LES model, you can enable the Werner-Wengle near-wall treatment through the following text command: `define/models/viscous/near-wall-treatment/werner-wengle-wall-fn?`. See LES Near-Wall Treatment in the Theory Guide for details.

11.14. Model Constants

It is also possible to modify the **Model Constants**, but this is not necessary for most applications. For more information about the constants, see Spalart-Allmaras Model through Large Eddy Simulation (LES) Model (in the Theory Guide). Note that **C1-PS** and **C2-PS** are the constants C_1 and C_2 in the linear pressure-strain approximation of Equation 4.209 and Equation 4.210 (in the Theory Guide), and **C1'-PS** and **C2'-PS** are the constants $C_1^{'}$ and $C_2^{'}$ in Equation 4.211 (in the Theory Guide). **C1-SSG-PS**, **C1'-SSG-PS**, **C2-SSG-PS**, **C3-SSG-PS**, **C3'-SSG-PS**, **C4-SSG-PS**, and **C5-SSG-PS** are the constants C_1, C_1^*, C_2, C_3, C_3^*, C_4, and C_5 in the quadratic pressure-strain approximation of Equation 4.220 (in the Theory Guide).

11.15. Setting Up the Embedded Large Eddy Simulation (ELES) Model

As described in Embedded Large Eddy Simulation (ELES) in the Theory Guide, the Embedded Large Eddy Simulation model is used when modeling a smaller embedded LES zone within a larger RANS computational domain. Recall that the interface between the upstream RANS zone and the LES zone must be defined (by assigning the interface to a velocity fluctuation algorithm). In addition, the interface between the LES zone and the downstream RANS zone must be considered.

This section describes how to set up the Embedded Large Eddy Simulation model.

1. In the **Viscous Model** dialog box, enable any RANS model (for example, k-ε, k-ω), or you can select DES, SAS, SDES, or SBES. The only RANS model not compatible with ELES is the Spalart-Allmaras model, as a one-equation model cannot provide the required turbulent length scale to the interface method.

   If a RANS model is selected, the model is applied globally to the computational domain, however, values for turbulence variables are frozen within the ELES region. The frozen state of the ELES zone is used to determine the flow conditions (for k-ε, k-ω, and so on) at the downstream LES-RANS zone interface. Note that this approach requires a fairly well converged global RANS solution to start with.

   If DES, SAS, SDES, or SBES is used in the outer zone, these models are not frozen, but run in the background in the ELES region, obtaining proper flow conditions (for k-ε, k-ω, etc.) at the downstream LES-RANS zone interface.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. For the specified fluid cell zone, enable **LES Zone** in the **Fluid** dialog box (Figure 11.21: Specifying an ELES Zone in the Fluid Dialog Box (p. 795)). This enables the **Embedded LES** tab in the **Fluid** dialog box.

   Note

   When DES, SAS, SDES, or SBES is used for the global model, this and the next step can, but need not, be skipped. (Remember the general limitations of DES in free, wall-independent flows.) If you choose to skip these steps, then proceed with step 4.

   The **ELES Zone** option will only appear in the **Fluid** dialog box if you select the **Transient Solver**. This must be done manually for RANS models.

3. In the **Embedded LES** tab, you can then specify the **Embedded Subgrid-Scale Model**, and the **Momentum Spatial Discretization**.

   For the **Embedded Subgrid-Scale Model**, the following subgrid-scale submodels are available (Subgrid-Scale Model (p. 803)):

   - **Smagorinsky-Lilly**
   - **WALE**
   - **Dynamic Smagorinsky** (which is the same as the LES model with **Smagorinsky-Lilly** selected and the **Dynamic Stress** option enabled)
   - **WMLES**
   - **WMLES S-Omega**

   The default **Embedded-Subgrid Scale Model** for the ELES model is the **WALE** model.

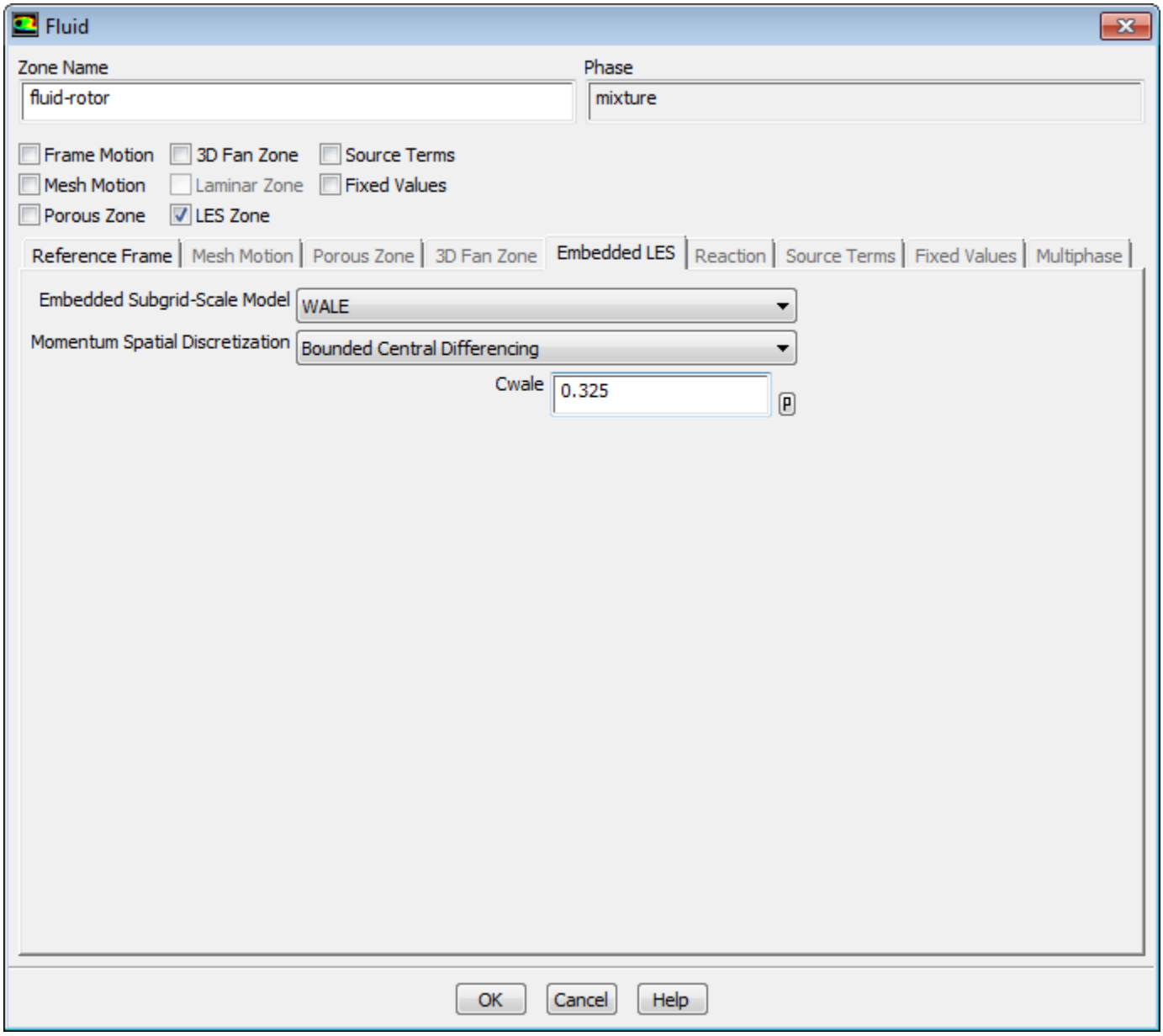
   When the WALE model is selected, you can specify a value for **Cwale** to define C_w (see Wall-Adapting Local Eddy-Viscosity (WALE) Model in the Theory Guide for details). Likewise, when the Smagorinsky-Lilly model is selected, you can specify a value for **Cs** to define C_s (see Smagorinsky-Lilly Model in the Theory Guide for details).

   For the **Momentum Spatial Discretization**, the following options are available:

   - **Bounded Central Differencing**
   - **Central Differencing**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

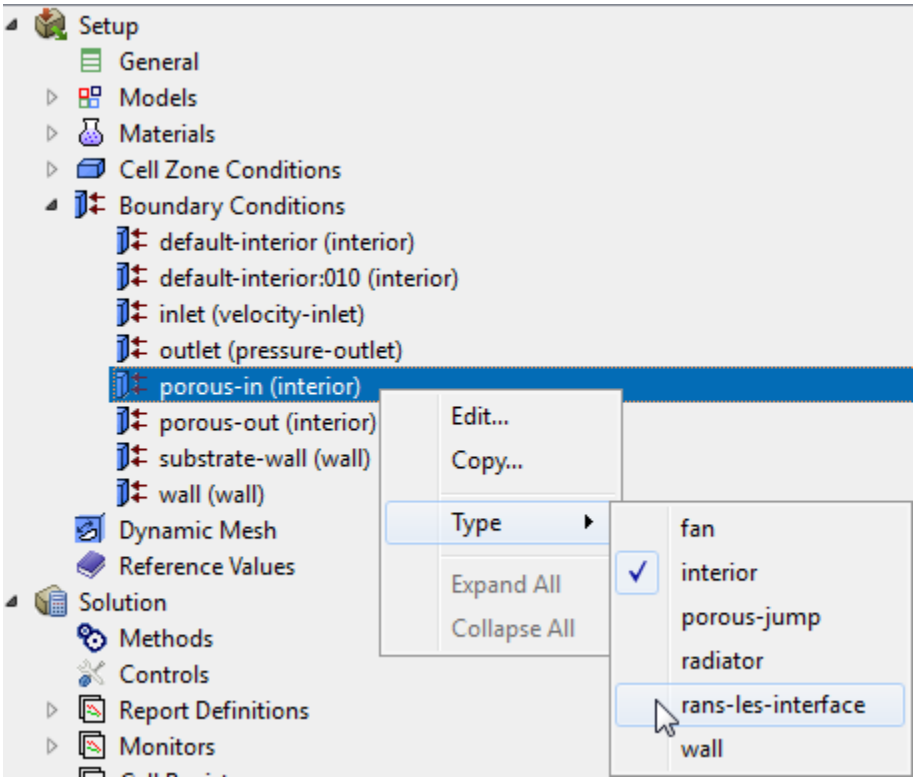
Figure 11.21: Specifying an ELES Zone in the Fluid Dialog Box



4. Select an appropriate interior interface and designate it as the RANS-LES interface, by right-clicking the boundary zone name in the tree (under **Setup/Boundary Conditions**) and selecting **rans-les-interface** in the **Type** sub-menu.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.22: Specifying the RANS/LES Interface



The **RANS/LES Interface** dialog box (Figure 11.23: The RANS/LES Interface Dialog Box (p. 797)) will open, allowing you to assign a **Zone Name**, as well as the **Fluctuating Velocity Algorithm** and the **Number Of Vortices**. The options for the **Fluctuating Velocity Algorithm** are:

- **No Perturbations**
- **Spectral Synthesizer**
- **Vortex Method**

For more information about these options, refer to Inlet Boundary Conditions for the LES Model in the Theory Guide.

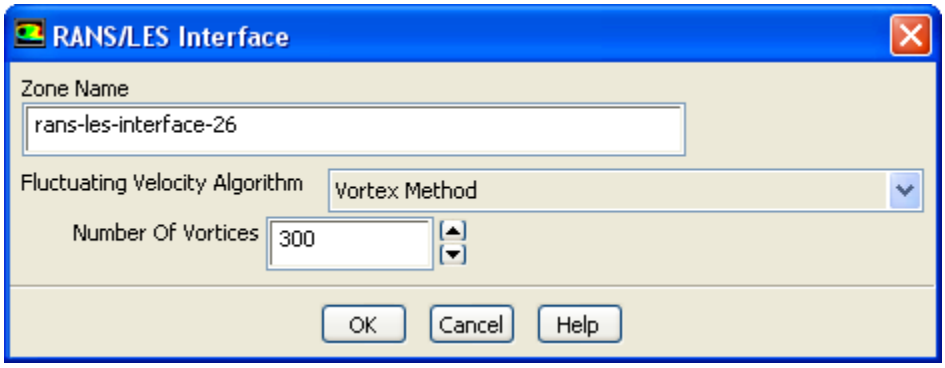
Note that the **Number Of Vortices** is the amount of vortices that the selected method distributes randomly over the face zone and uses to generate turbulent fluctuations. The value should be large enough to make sure there are no spots on the face zone that are unaffected by any vortex. Large numbers may slightly increase the CPU effort, but will not impair the results.

Note

The minimum advised number of vortices is approximately a quarter of the number of cell faces at the LES side of the interface.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.23: The RANS/LES Interface Dialog Box



If the RANS/LES interface is a non-conformal mesh interface, you can find the name of the interface interior zone that you want to change into a **rans-les-interface** zone by using the **List** button in the Mesh Interfaces Dialog Box (p. 2732).

Important

It is recommended that the RANS-LES interface be situated in a region where there is no backflow.

11.16. Setup Options for All Turbulence Modeling

For more information about the various options available for the turbulence models, see Spalart-Allmaras Model through Large Eddy Simulation (LES) Model (in the Theory Guide). Instructions for activating these options are provided here.

For additional information, see the following sections:

11.16.1. Including the Viscous Heating Effects
11.16.2. Including Turbulence Generation Due to Buoyancy
11.16.3. Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models
11.16.4. Including the Compressibility Effects Option
11.16.5. Including Production Limiters for Two-Equation Models
11.16.6. Including the Intermittency Transition Model
11.16.7. Vorticity- and Strain/Vorticity-Based Production
11.16.8. Delayed Detached Eddy Simulation (DDES)
11.16.9. Differential Viscosity Modification
11.16.10. Swirl Modification
11.16.11. Low-Re Corrections
11.16.12. Shear Flow Corrections
11.16.13. Turbulence Damping
11.16.14. Including Pressure Gradient Effects
11.16.15. Including Thermal Effects
11.16.16. Including the Wall Reflection Term
11.16.17. Solving the k Equation to Obtain Wall Boundary Conditions
11.16.18. Quadratic Pressure-Strain Model
11.16.19. Stress-Omega and Stress-BSL Models
11.16.20. Subgrid-Scale Model
11.16.21. Customizing the Turbulent Viscosity
11.16.22. Customizing the Turbulent Prandtl and Schmidt Numbers

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.16.23. Modeling Turbulence with Non-Newtonian Fluids
11.16.24. Including Scale-Adaptive Simulation with ω-Based URANS Models
11.16.25. Including Detached Eddy Simulation with the Transition SST Model
11.16.26. Including the SDES or SBES Model with BSL, SST, and Transition SST Models
11.16.27. Shielding Functions for the BSL / SST / Transition SST Detached Eddy Simulation Model

11.16.1. Including the Viscous Heating Effects

For information about including viscous heating effects in your model, see Inclusion of the Viscous Dissipation Terms in the Theory Guide and Solving Heat Transfer Problems (p. 825).

11.16.2. Including Turbulence Generation Due to Buoyancy

If you specify a nonzero gravity force (in the Operating Conditions Dialog Box (p. 2382)), and you are modeling a non-isothermal flow, the generation of turbulent kinetic energy due to buoyancy (G_b in Equation 4.39) is, by default, always included in the k equation. However, ANSYS Fluent does not, by default, include the buoyancy effects on ε.

To include the buoyancy effects on ε, you must turn on the **Full Buoyancy Effects** option under **Options** in the Viscous Model Dialog Box (p. 2176).

This option is available for all three k-ε models and for the ε-based RSM (that is, with the **Linear Pressure-Strain** or **Quadratic Pressure-Strain** model selected).

11.16.3. Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models

Eddy-viscosity models display an insensitivity to streamline curvature and system rotation, which play a significant role in many turbulent flows of practical interest, as described in Curvature Correction for the Spalart-Allmaras and Two-Equation Models in the Theory Guide. A modification to the turbulence production term is available to sensitize the following standard eddy-viscosity models to the effects of streamline curvature and system rotation:

- Spalart-Allmaras one-equation model
- Standard, RNG, and Realizable (k-ε) models
- Standard (k-ω), BSL, SST, and Transition SST
- Scale-Adaptive Simulation (SAS)
- Detached Eddy Simulation with BSL (DES-BSL), with SST (DES-SST), with Spalart-Allmaras (DES-SA), and with Realizable (k-ε) model (DES-rke).
- Shielded Detached Eddy Simulation (SDES)
- Stress-Blended Eddy Simulation (SBES)

Note that both the RNG and Realizable (k-ε) turbulence models already have their own terms to include rotational or swirl effects (see RNG Swirl Modification and Modeling the Turbulent Viscosity in the Theory Guide for more information). The curvature correction option should therefore be used with caution for these two models and is offered mainly for completeness for RNG and Realizable (k-ε).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Enable the **Curvature Correction** option under **Options** in the Viscous Model Dialog Box (p. 2176).

11.16.4. Including the Compressibility Effects Option

The **Compressibility Effects** option is available under **Options** in the Viscous Model Dialog Box (p. 2176) for most ε-based models, ω-based models, and Reynolds stress models when the compressible form of the ideal gas law or the real-gas model is enabled. This option can improve the prediction of free shear layers at high Mach numbers. See Model Enhancements (p. 755) for recommendations on when to use this option. For details about how this correction is implemented, see Effects of Compressibility on Turbulence in the k-ε Models and Compressibility Effects in the Theory Guide.

11.16.5. Including Production Limiters for Two-Equation Models

A disadvantage of standard two-equation turbulence models is the excessive generation of the turbulence energy, G_k, in the vicinity of stagnation points. In order to avoid the buildup of turbulent kinetic energy in the stagnation regions, two formulations are available in order to limit the production term in the turbulence kinetic energy equations:

- Production Limiter (for details, see Equation 4.382 in the *Fluent Theory Guide*)
- Production Kato-Launder (for details, see Equation 4.386 in the *Fluent Theory Guide*)

Both these formulations can be accessed under **Options** in the **Viscous Model** dialog box. The **Production Limiter** model coefficient C_{lim} is called the **Production Limiter Clip Factor** and has a default value of 10. This value can be modified under **Model Constants** once the **Production Limiter** formulation is enabled.

You can use the following text commands to set the formulations:

- Production Limiter

```
/define/models/viscous/turbulence-expert/production-limiter?[yes]
```

- Production Kato-Launder

```
/define/models/viscous/turbulence-expert/kato-launder-model?[yes]
```

The **Production Limiter** is available for the following turbulence models:

- Standard, RNG, and Realizable (k-ε) models
- Standard (k-ω), BSL, SST, and Transition SST
- Scale-Adaptive Simulation (SAS)
- Detached Eddy Simulation with BSL (DES-BSL), with SST (DES-SST), and with Realizable (k-ε) models (DES-rke)
- Shielded Detached Eddy Simulation (SDES)
- Stress-Blended Eddy Simulation (SBES)

By default, this limiter is enabled for all turbulence models based on the ω equation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Production Kato-Launder** formulation for the production term is available for the following turbulence models:

- Standard, and RNG (k-ε) models
- Standard (k-ω), BSL, SST, and Transition SST
- Scale-Adaptive Simulation (SAS)
- Detached Eddy Simulation with BSL (DES-BSL) and with SST (DES-SST)
- Shielded Detached Eddy Simulation (SDES)
- Stress-Blended Eddy Simulation (SBES)

The Kato-Launder formulation is enabled by default for the Transition SST model only.

11.16.6. Including the Intermittency Transition Model

The Intermittency Transition model is available for the BSL k-ω model, SST k-ω model, Scale-Adaptive Simulation with BSL / SST, Detached-Eddy Simulation with BSL / SST, Shielded Detached Eddy Simulation (SDES) with BSL / SST, and Stress-Blended Eddy Simulation (SBES) with BSL / SST. It can be included by enabling **Intermittency Transition Model** in the **Options** group box in the **Viscous Model** dialog box with the appropriate turbulence model selected.

When the **Intermittency Transition Model** option is enabled, the additional option **Include Crossflow Transition** is available. It allows you to include the effects of crossflow instability.

For details about the Intermittency Transition model, see Intermittency Transition Model in the Theory Guide.

11.16.7. Vorticity- and Strain/Vorticity-Based Production

For the Spalart-Allmaras model, you can choose either **Vorticity-Based Production** or **Strain/Vorticity-Based Production** under **Spalart-Allmaras Production** in the Viscous Model Dialog Box (p. 2176). If you choose **Vorticity-Based Production**, ANSYS Fluent will compute the value of the deformation tensor S using Equation 4.22 (in the Theory Guide); if you choose **Strain/Vorticity-Based Production**, it uses Equation 4.24 (in the Theory Guide).

(These options will not appear unless you have enabled the Spalart-Allmaras model.)

11.16.8. Delayed Detached Eddy Simulation (DDES)

The **Delayed DES** option is recommended when using the DES model. This option preserves the RANS model throughout the boundary layer. For more information, see Detached Eddy Simulation (DES) in the Theory Guide.

11.16.9. Differential Viscosity Modification

The RNG turbulence model in ANSYS Fluent has an option of using a differential formula for the effective viscosity μ_{eff} (Equation 4.44 in the Theory Guide) to account for the low-Reynolds-number effects. To

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

enable this option, the **Differential Viscosity Model** option under **RNG Options** in the Viscous Model Dialog Box (p. 2176) must be enabled.

Important

This option appears when you have enabled the RNG k-ε model.

11.16.10. Swirl Modification

After you have chosen the RNG model, the swirl modification takes effect, by default, for all three-dimensional flows and axisymmetric flows with swirl. The default swirl constant (α_s in Equation 4.46 in the Theory Guide) is set to 0.07, which works well for weakly to moderately swirling flows. However, for strongly swirling flows, you may need to use a larger swirl constant.

To change the value of the swirl constant, you must first enable the **Swirl Dominated Flow** option under **RNG Options** in the Viscous Model Dialog Box (p. 2176).

Important

This option will not appear unless you have enabled the RNG k-ε model.

11.16.11. Low-Re Corrections

If any of the k-ω models or the Stress-Omega RSM is used, you may enable a low-Reynolds-number correction to the turbulent viscosity by enabling the **Low-Re Corrections** option under **k-omega Options** in the **Viscous Model** dialog box. By default, this option is not enabled, and the damping coefficient (α^* in Equation 4.74 in the Theory Guide) is equal to 1.

11.16.12. Shear Flow Corrections

In the standard k-ω model and the Stress-Omega RSM, you also have the option of including corrections to improve the accuracy in predicting free shear flows. The **Shear Flow Corrections** option under the **k-omega Options** is enabled by default in the **Viscous Model** dialog box, as these corrections are included in the standard k-ω model [130] (p. 2894). When this option is enabled, ANSYS Fluent will calculate f_β^* using Equation 4.84 (in the Theory Guide) and f_β using Equation 4.91 (in the Theory Guide). If this option is disabled, f_β^* and f_β will be set equal to 1.

11.16.13. Turbulence Damping

In the standard, BSL, or SST k-ω model, you have the option of including turbulence damping, which is required for the accurate modeling of the interfacial area. The **Turbulence Damping** option under **k-omega Options** is available for the VOF and Mixture models and also available with the Eulerian multiphase model when using the immiscible fluid model. When this option is enabled, you can set the **Damping Factor**, which by default is set to 10. For a theoretical discussion about turbulence damping, refer to Turbulence Damping.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.16.14. Including Pressure Gradient Effects

If the enhanced wall treatment is used, you may include the effects of pressure gradients by enabling the **Pressure Gradient Effects** option under the **Enhanced Wall Treatment Options**. When this option is enabled, ANSYS Fluent will include the coefficient α in Equation 4.348 (in the Theory Guide).

11.16.15. Including Thermal Effects

If the enhanced wall treatment is used, you may include thermal effects by enabling the **Thermal Effects** option under **Enhanced Wall Treatment Options**. When this option is enabled, ANSYS Fluent will include the coefficient β in Equation 4.348 (in the Theory Guide). γ will also be included in Equation 4.348 when the **Thermal Effects** option is enabled if the ideal gas law is selected for the fluid density in the **Create/Edit Materials** dialog box.

11.16.16. Including the Wall Reflection Term

If the RSM is used with the default model for linear pressure-strain, ANSYS Fluent will, by default, include the wall-reflection effects in the pressure-strain term. That is, ANSYS Fluent will calculate $\phi_{ij,w}$ using Equation 4.211 (in the Theory Guide) and include it in Equation 4.208 (in the Theory Guide). For the quadratic pressure-strain model, Stress-Omega model, and Stress-BSL model, the wall-reflection effects are not required and are not included.

Important

The empirical constants and the function f used in the calculation of $\phi_{ij,w}$ are calibrated for simple canonical flows such as channel flows and flat-plate boundary layers involving a single wall. If the flow involves multiple walls and the wall has significant curvature (for example, an axisymmetric pipe or curvilinear duct), the inclusion of the wall-reflection term in Equation 4.211 (in the Theory Guide) may not improve the accuracy of the RSM predictions. In such cases, you can disable the wall-reflection effects by turning off the **Wall Reflection Effects** under **Reynolds-Stress Options** in the Viscous Model Dialog Box (p. 2176).

11.16.17. Solving the k Equation to Obtain Wall Boundary Conditions

For the ε-based Reynolds stress models, by default ANSYS Fluent uses the explicit setting of boundary conditions for the Reynolds stresses near the walls, with the values computed with Equation 4.238 (in the Theory Guide). The turbulent kinetic energy, k, is calculated by solving the k equation obtained by summing Equation 4.204 (in the Theory Guide) for normal stresses. To disable this option and use the wall boundary conditions given in Equation 4.239 (in the Theory Guide), turn off the **Wall BC from k Equation** under the **Reynolds-Stress Options** in the **Viscous Model** dialog box.

Important

This option will not appear unless you have enabled the **Reynolds Stress** model.

11.16.18. Quadratic Pressure-Strain Model

To use the quadratic pressure-strain model described in Quadratic Pressure-Strain Model (in the Theory Guide), enable the **Quadratic Pressure-Strain Model** option under **Reynolds-Stress Options** in the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Viscous Model Dialog Box (p. 2176). (This option will not appear unless you have enabled the RSM.) The following options are not available when the **Quadratic Pressure-Strain Model** is enabled:

- **Wall Reflection Effects** under **Reynolds-Stress Options**
- **Enhanced Wall Treatment** under **Near-Wall Treatment**

11.16.19. Stress-Omega and Stress-BSL Models

To use the Stress-Omega model (described in Stress-Omega Model) or the Stress-BSL model (described in Stress-BSL Model), select **Stress-Omega** or **Stress-BSL**, respectively, from the **Reynolds-Stress Model** list in the Viscous Model Dialog Box (p. 2176). (These options will not appear unless you have enabled the RSM.) The following options are not available when the **Stress-Omega** or the **Stress-BSL** model is selected:

- **Wall BC from k Equation** under **Reynolds-Stress Options**
- **Wall Reflection Effects** under **Reynolds-Stress Options**
- **Standard Wall Functions** under **Near-Wall Treatment**
- **Scalable Wall Functions** under **Near-Wall Treatment**
- **Non-Equilibrium Wall Functions** under **Near-Wall Treatment**
- **Enhanced Wall Treatment** under **Near-Wall Treatment**

Instead, the following options are available:

- **Low-Re Corrections** under **k-omega Options** (for **Stress-Omega** only)
- **Shear Flow Corrections** under **k-omega Options** (for **Stress-Omega** only)
- **Scale-Adaptive Simulation (SAS)** under **Scale-Resolving Simulation Options**

11.16.20. Subgrid-Scale Model

If you have selected the **Large Eddy Simulation** model, you will be able to choose one of the subgrid-scale models described in Subgrid-Scale Models (in the Theory Guide). You can choose from the **Smagorinsky-Lilly**, **WALE**, **WMLES**, **WMLES S-Omega**, or **Kinetic-Energy Transport** subgrid-scale models. Note that **Dynamic Stress** is an option available with the **Smagorinsky-Lilly** model (and when it is enabled, this model is referred to as the "dynamic Smagorinsky" model), while the **Kinetic-Energy Transport** model is always run as a dynamic model. The **Dynamic Fvar** option is available for all of the subgrid-scale models when **Non-Premixed Combustion** or **Partially Premixed Combustion** is selected in the Species Model Dialog Box (p. 2216).

Important

These options will not appear unless you have enabled the LES model.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.16.21. Customizing the Turbulent Viscosity

If you are using the Spalart-Allmaras, k-ε, k-ω, DES, or LES models, a UDF can be used to customize the turbulent viscosity. This option will enable you to modify μ_t in the case of the Spalart-Allmaras, k-ε, and k-ω models, and incorporate completely new subgrid models in the case of the LES model. SBES with BSL / SST allows the usage of a UDF in order to provide a custom formulation for μ_t in the RANS region, but not for the subgrid model in the LES region. More information about UDFs can be found in the Fluent Customization Manual.

In the Viscous Model Dialog Box (p. 2176), under **User-Defined Functions**, select the appropriate user-defined function in the **Turbulent Viscosity** drop-down list. For the LES model, select the appropriate UDF in the **Subgrid-Scale Turbulent Viscosity** drop-down list.

11.16.22. Customizing the Turbulent Prandtl and Schmidt Numbers

You can use UDFs to customize the turbulent Prandtl and Schmidt numbers. For the standard and realizable k-ε models and the standard k-ω model, your UDF can calculate the TKE Prandtl number (σ_k) and either the TDR or SDR Prandtl number (σ_ε or σ_ω), depending on whether you are using the k-ε or k-ω model. All turbulence models allow you to use UDFs for the turbulent Prandtl number at the wall (Pr_t in Equation 4.319 in the Theory Guide), and all turbulent models except for the RNG k-ε model allow you to use UDFs for the turbulent Prandtl number for energy (Pr_t in, for example, Equation 4.63 or Equation 4.229 in the Theory Guide), as well as the turbulent Schmidt number (Sc_t, in Equation 7.3 in the Theory Guide) if species transport has been enabled. Note that UDFs cannot be used for the turbulent Prandtl number for energy or the turbulent Schmidt number if you have enabled **Dynamic Energy Flux** or **Dynamic Scalar Flux**, respectively, as part of the LES model. More information about UDFs can be found in the Fluent Customization Manual.

In the Viscous Model Dialog Box (p. 2176), under **User-Defined Functions**, select the appropriate UDF from the drop-down lists in the **Prandtl Numbers** or **Prandtl and Schmidt Numbers** group box. The options will depend on the turbulence model you have selected (as noted previously), and include: **TKE Prandtl Number**, **TDR Prandtl Number**, **SDR Prandtl Number**, **Energy Prandtl Number**, **Wall Prandtl Number**, and **Turbulent Schmidt Number**. Note that the availability of these drop-down lists can be affected by your settings for the energy equation and the multiphase and combustion models.

11.16.23. Modeling Turbulence with Non-Newtonian Fluids

If the turbulent flow involves non-Newtonian fluids, you can use the `define/models/viscous/turbulence-expert/turb-non-newtonian?` text command to enable the selection of non-Newtonian options for the material viscosity. See Viscosity for Non-Newtonian Fluids (p. 492) for details about these options.

11.16.24. Including Scale-Adaptive Simulation with ω-Based URANS Models

You have the option of including Scale-Adaptive Simulation (SAS) with most ω-based URANS models. For details about SAS and how to include it, see Scale-Adaptive Simulation (SAS) (p. 758) and Setting Up Scale-Adaptive Simulation (SAS) Modeling (p. 781), respectively.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.16.25. Including Detached Eddy Simulation with the Transition SST Model

You have the option of including Detached Eddy Simulation (DES) with the Transition SST model. For details about DES and how to include it, see Detached Eddy Simulation (DES) (p. 758) and Setting Up DES with the Transition SST Model (p. 789), respectively.

11.16.26. Including the SDES or SBES Model with BSL, SST, and Transition SST Models

To enable a hybrid RANS-LES model with an improved shielding function compared to DDES / IDDES, you can enable the **Stress Blending (SBES) / Shielded DES** option in the **Scale-Resolving Simulation Options** group box of the **Viscous Model** dialog box. This option is available for 3D transient cases when you have selected one of the following models (see Figure 11.24: The Viscous Model Dialog Box with the SBES Options (p. 806)):

- Baseline (BSL) k-ω model
- Shear-Stress Transport (SST) k-ω model
- Transition SST model

You can then select one of the following from the **Hybrid Model** list:

- **SDES** to apply only a shielding function (Shielded Detached Eddy Simulation (SDES) in the *Fluent Theory Guide*)
- **SBES** to apply a shielding function with stress blending (Stress-Blended Eddy Simulation (SBES) in the *Fluent Theory Guide*)
- **SBES with User-Defined Function** to define your own blending function zonally for cases where the division between RANS and LES is clear from the geometry and the flow physics (`DEFINE_SBES_BF` in the *Fluent Customization Manual*)

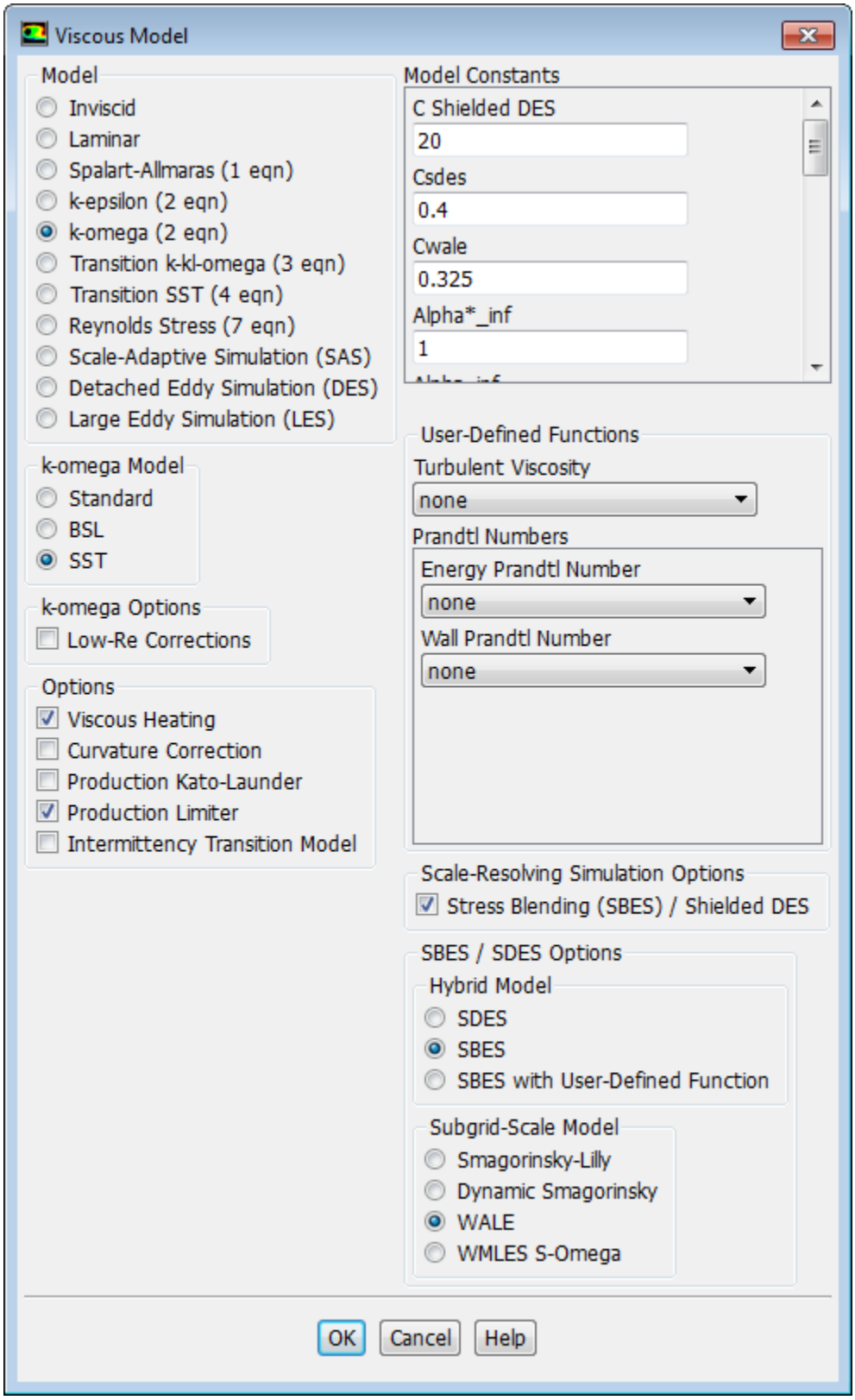
If you select **SBES** or **SBES with User-Defined Function** for the **Hybrid Model**, you can also select one of the following subgrid-scale (LES) models:

- **Smagorinsky-Lilly** model
- **Dynamic Smagorinsky**-Lilly model
- Wall-Adapting Local Eddy-Viscosity (**WALE**) model
- Algebraic Wall-Modeled LES S-Ω (**WMLES S-Omega**) model

The **WALE** model is recommended, as it provides the lowest eddy viscosity in 2D flow regions and thereby allows the flow to quickly develop 3D turbulence structures in separating shear layers. The combination of SBES with the **WMLES S-Omega** formulation is likely not a useful combination, as the SBES-WALE model combination can by itself act in a WMLES mode.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.24: The Viscous Model Dialog Box with the SBES Options



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.16.27. Shielding Functions for the BSL / SST / Transition SST Detached Eddy Simulation Model

For the DES model with the BSL k-ω model, SST k-ω model, or the Transition SST model, the shielding (or blending) functions F1 and F2 are defined in Equation 4.107 and Equation 4.118 in the Theory Guide, respectively. Note that the F2 function is more conservative than the F1 function. In addition, the shielding functions DDES (Delayed DES) and IDDES (Improved Delayed DES), are described in DES with the BSL or SST k-ω Model and Improved Delayed Detached Eddy Simulation (IDDES) in the Theory Guide. DDES is recommended and is used as the default.

11.17. Defining Turbulence Boundary Conditions

For additional information, see the following sections:

11.17.1. Wall Roughness Effects
11.17.2. The Spalart-Allmaras Model
11.17.3. k-ε Models and k-ω Models
11.17.4. Reynolds Stress Model
11.17.5. Large Eddy Simulation Model

11.17.1. Wall Roughness Effects

You may want to include the effects of the wall roughness on selected wall boundaries as part of your turbulent simulation. In such cases, you can specify the roughness parameters (roughness height and roughness constant) in the dialog boxes of the corresponding wall boundaries (see Setting the Roughness Parameters (p. 368)).

Note

Rough walls cannot be used together with the following model combinations:

- an ε-equation model with enhanced wall treatment or the Menter-Lechner near-wall treatment

  Note that the following are the relevant ε-equation models:

  – all of the k-ε models (that is, standard, RNG, and realizable)

  – the Reynolds stress model with the **Linear Pressure-Strain** model selected

  – the detached eddy simulation (DES) model with the **Realizable k-epsilon** option selected

- the Reynolds stress model with the **Stress-Omega** or **Stress-BSL** model selected
- the transition k-kl-ω model
- the large eddy simulation (LES) model

11.17.2. The Spalart-Allmaras Model

When you are modeling turbulent flows in ANSYS Fluent using the Spalart-Allmaras model, you must provide the boundary conditions for $\tilde{\nu}$ in addition to other mean solution variables. The boundary conditions for $\tilde{\nu}$ at the walls are internally taken care of by ANSYS Fluent, which obviates the need for

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

your inputs. The boundary condition input for $\tilde{\nu}$, which you must enter in ANSYS Fluent, is the one at inlet boundaries (for example, velocity inlets, pressure inlets). In many situations, it is important to specify correct or realistic boundary conditions at the inlets, because the inlet turbulence can significantly affect the downstream flow.

11.17.3. k-ε Models and k-ω Models

When you are modeling turbulent flows in ANSYS Fluent using one of the k-ε models or one of the k-ω models, you must provide the boundary conditions for k and ε (or k and ω) in addition to other mean solution variables. The boundary conditions for k and ε (or k and ω) at the walls are internally taken care of by ANSYS Fluent, which obviates the need for your inputs. The boundary condition inputs for k and ε (or k and ω), which you must enter in ANSYS Fluent, are the ones at inlet boundaries (for example, velocity inlets, pressure inlets). In many situations, it is important to specify correct or realistic boundary conditions at the inlets, because the inlet turbulence can significantly affect the downstream flow.

See Determining Turbulence Parameters (p. 303) for details about specifying the boundary conditions for k and ε (or k and ω) at the inlets.

Additionally, you can control whether or not to set the turbulent viscosity to zero within a laminar zone. If the fluid zone in question is laminar, the text command `define/ boundary-conditions/fluid` will contain an option called `Set Turbulent Viscosity to zero within laminar zone?`. By setting this option to `yes`, ANSYS Fluent will set both the production term in the turbulence transport equation and μ_t to zero. In contrast, when the **Laminar Zone** option is enabled in a **Fluid** cell zone condition dialog box, only the production term is set to zero. See Specifying a Laminar Zone (p. 255) for details about laminar zones.

Important

Note that the laminar zone feature is also available for the Spalart-Allmaras and Reynolds stress models.

11.17.4. Reynolds Stress Model

The specification of turbulent boundary conditions for the RSM is the same as for the other turbulence models for all boundaries except at boundaries where flow enters the domain. Additional input methods are available for these boundaries and are described here.

When you choose to use the RSM, the default inlet boundary condition inputs required are identical to those required when the k-ε model is active. You can input the turbulence quantities using any of the turbulence specification methods described in Determining Turbulence Parameters (p. 303). ANSYS Fluent then uses the specified turbulence quantities to derive the Reynolds stresses at the inlet from the assumption of isotropy of turbulence:

$$\overline{u_i'^2} = \frac{2}{3}k \quad (i=1,2,3) \tag{11.6}$$

$$\overline{u'_i u'_j} = 0.0 \tag{11.7}$$

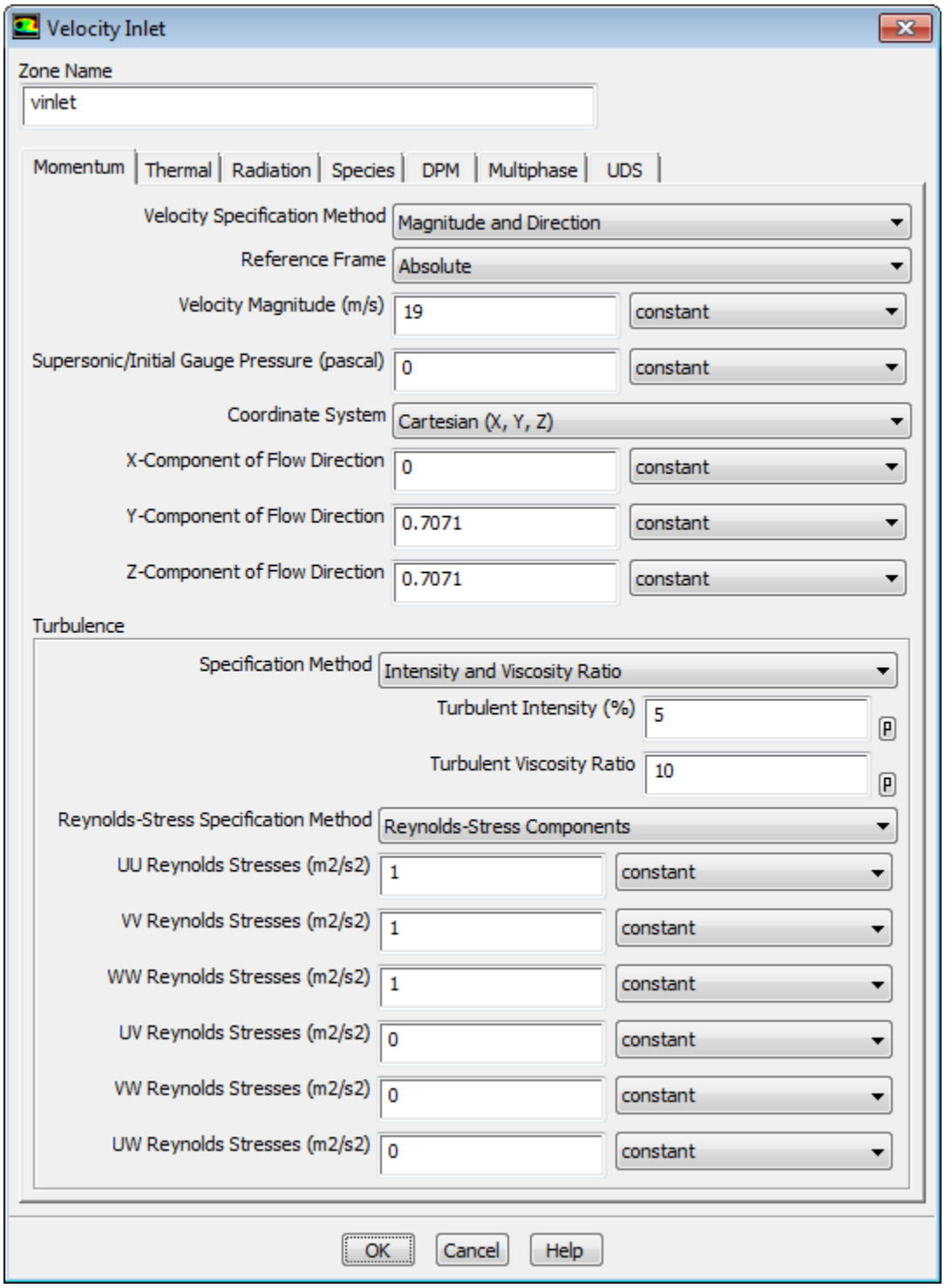
where $\overline{u_i'^2}$ is the normal Reynolds stress component in each direction. The boundary condition for ε is determined in the same manner as for the k-ε turbulence models (see Determining Turbulence Parameters (p. 303)). To use this method, you will select **K or Turbulent Intensity** as the **Reynolds-Stress Specification Method** in the appropriate boundary condition dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Alternately, you can directly specify the Reynolds stresses by selecting the **Reynolds-Stress Components** as the **Reynolds-Stress Specification Method** in the boundary condition dialog box. When this option is selected, you should input the Reynolds stresses directly.

You can set the Reynolds stresses by using constant values, profile functions of coordinates (see Profiles (p. 435)), or user-defined functions (in the Fluent Customization Manual).

Figure 11.25: Specifying Inlet Boundary Conditions for the Reynolds Stresses



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.17.5. Large Eddy Simulation Model

It is possible to specify the magnitude of random fluctuations of the velocity components at an inlet only if the velocity inlet boundary condition is selected. In this case, you must specify a **Turbulence Intensity** that determines the magnitude of the random perturbations on individual mean velocity components as described in Inlet Boundary Conditions for the LES Model (in the Theory Guide). For all boundary types other than velocity inlets, the boundary conditions for LES remain the same as for laminar flows.

11.18. Providing an Initial Guess for k and ε (or k and ω)

For flows using one of the k-ε models, one of the k-ω models, or the RSM, the converged solutions or (for unsteady calculations) the solutions after a sufficiently long time has elapsed should be independent of the initial values for k and ε (or k and ω). For better convergence, however, it is beneficial to use a reasonable initial guess for k and ε (or k and ω).

In general, it is recommended that you start from a fully-developed state of turbulence. When you use a y^+-insensitive wall treatment, it is critically important to specify fully-developed turbulence fields. Guidelines are provided below.

- If you were able to specify reasonable boundary conditions at the inlet, it may be a good idea to compute the initial values for k and ε (or k and ω) in the whole domain from these boundary values. (See Initializing the Solution (p. 1591) for details.)

- For more complex flows (for example, flows with multiple inlets with different conditions) it may be better to specify the initial values in terms of turbulence intensity. 5–10% is enough to represent fully-developed turbulence. The values of k can then be computed from the turbulence intensity and the characteristic mean velocity magnitude of your problem ($k=1.5\left(Iu_{avg}\right)^2$).

  You should specify an initial guess for ε so that the resulting eddy viscosity ($C_\mu\frac{k^2}{\varepsilon}$) is sufficiently large in comparison to the molecular viscosity. In fully-developed turbulence, the turbulent viscosity is roughly two orders of magnitude larger than the molecular viscosity. From this, you can compute ε.

$$\varepsilon=C_\mu\frac{k^2}{\mu_T}=C_\mu\frac{k^2}{\frac{\mu_T}{\mu}\mu} \tag{11.8}$$

  where $\frac{\mu_T}{\mu}$ is the turbulent viscosity ratio which can be prescribed at the inlet and then used for the domain initialization.

  Note that, for the RSM, Reynolds stresses are initialized automatically using Equation 11.6 (p. 808) and Equation 11.7 (p. 808).

11.19. Solution Strategies for Turbulent Flow Simulations

Compared to laminar flows, simulations of turbulent flows are more challenging in many ways. For the Reynolds-averaged approach, additional equations are solved for the turbulence quantities. Since the equations for mean quantities and the turbulent quantities (μ_t, k, ε, ω, or the Reynolds stresses) are strongly coupled in a highly nonlinear fashion, it takes more computational effort to obtain a converged turbulent solution than to obtain a converged laminar solution. The LES model, while embodying a simpler, algebraic model for the subgrid-scale viscosity, requires a transient solution on a very fine mesh.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The fidelity of the results for turbulent flows is largely determined by the turbulence model being used. Here are some guidelines that can enhance the quality of your turbulent flow simulations.

For additional information, see the following sections:

11.19.1. Mesh Generation
11.19.2. Accuracy
11.19.3. Convergence
11.19.4. RSM-Specific Solution Strategies
11.19.5. LES-Specific Solution Strategies

11.19.1. Mesh Generation

The following are suggestions to follow when generating the mesh for use in your turbulent flow simulation:

- First imagine the flow under consideration, then identify the main flow features expected in the flow using your physical intuition or any data for a similar flow situation. Generate a mesh that can resolve the major features that you expect.
- If the flow is wall-bounded and the wall is expected to affect the flow significantly, you should take additional care when generating the mesh. See Grid Resolution for RANS Models (p. 756) and Wall Boundary Layers (p. 760) for guidelines.

11.19.2. Accuracy

The suggestions below are provided to help you obtain better accuracy in your results:

- Use the turbulence model that is better suited for the salient features you expect to see in the flow (see Choosing a Turbulence Model (p. 751)).
- Because the mean quantities have larger gradients in turbulent flows than in laminar flows, it is recommended that you use high-order schemes for the convection terms. This is especially true if you employ a triangular or tetrahedral mesh. Note that excessive numerical diffusion adversely affects the solution accuracy, even with the most elaborate turbulence model.
- In some flow situations involving inlet boundaries, the flow downstream of the inlet is dictated by the boundary conditions at the inlet. In such cases, you should exercise care to make sure that reasonably realistic boundary values are specified.

11.19.3. Convergence

The suggestions below are provided to help you enhance convergence for turbulent flow calculations:

- Starting with excessively crude initial guesses for mean and turbulence quantities may cause the solution to diverge. A safe approach is to start your calculation using conservative (small) under-relaxation parameters and (for the density-based solvers) a conservative Courant number, and increase them gradually as the iterations proceed and the solution begins to settle down.
- It is also helpful for faster convergence to start with reasonable initial guesses for the k and ε (or k and ω) fields. Particularly when a y^+-insensitive wall treatment is used, it is important to start with a sufficiently developed turbulence field, as recommended in Providing an Initial Guess for k and ε (or k and ω) (p. 810), to avoid the need for an excessive number of iterations to develop the turbulence field.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When you are using the RNG k-ε model, an approach that might help you achieve better convergence is to obtain a solution with the standard k-ε model before switching to the RNG model. Due to the additional nonlinearities in the RNG model, lower under-relaxation factors and (for the density-based solvers) a lower Courant number might also be necessary.

Note that when you use the enhanced wall treatment (EWT-ε), you may sometimes find during the calculation that the residual for ε is reported to be zero. This happens when your flow is such that Re_y is less than 200 in the entire flow domain, and ε is obtained from the algebraic formula (Equation 4.342 in the Theory Guide) instead of from its transport equation.

11.19.4. RSM-Specific Solution Strategies

Using the RSM creates a high degree of coupling between the momentum equations and the turbulent stresses in the flow, and therefore the calculation can be more prone to stability and convergence difficulties than with the k-ε models. When you use the RSM, therefore, you may need to adopt special solution strategies in order to obtain a converged solution. The following strategies are generally recommended:

- Begin the calculations using the standard k-ε model. Turn on the RSM and use the k-ε solution data as a starting point for the RSM calculation.

- Use low under-relaxation factors (0.2 to 0.3) and (for the density-based solvers) a low Courant number for highly swirling flows or highly complex flows. In these cases, you may need to reduce the under-relaxation factors both for the velocities and for all of the stresses.

Instructions for setting these solution parameters are provided below. If you are applying the RSM to prediction of a highly swirling flow, you will want to consider the solution strategies discussed in Swirling and Rotating Flows (p. 587) as well.

11.19.4.1. Under-Relaxation of the Reynolds Stresses

ANSYS Fluent applies under-relaxation to the Reynolds stresses. You can set under-relaxation factors using the Solution Controls Task Page (p. 2508).

Solution → Controls

The default settings of 0.5 are recommended for most cases. You may be able to increase these settings and speed up the convergence when the RSM solution begins to converge.

In some situations, when poor convergence is observed one might facilitate the convergence rates by modifying some of the under-relaxation values whilst leaving the others unchanged. This might be a more successful approach than the simple scaling of all under-relaxation values.

11.19.4.2. Disabling Calculation Updates of the Reynolds Stresses

In some instances, you may want to let the current Reynolds stress field remain fixed, skipping the solution of the Reynolds transport equations while solving the other transport equations. You can enable/disable all Reynolds stress equations in the **Equations** dialog box.

Solution → Controls → Equations...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.19.4.3. Residual Reporting for the RSM

When you use the RSM for turbulence, ANSYS Fluent reports the equation residuals for the individual Reynolds stress transport equations. You can apply the usual convergence criteria to the Reynolds stress residuals: normalized residuals in the range of 10^{-3} usually indicate a practically-converged solution. However, you may need to apply tighter convergence criteria (below 10^{-4}) to ensure full convergence.

11.19.5. LES-Specific Solution Strategies

Large eddy simulation involves running a transient solution from some initial condition, on an appropriately fine mesh, using an appropriate time step size. The solution must be run long enough to become independent of the initial condition and to enable the statistics of the flow field to be determined.

The following are suggestions to follow when running a large eddy simulation:

1. Start by running a steady-state flow simulation using a Reynolds-averaged turbulence model such as standard k-ε, k-ω, or even RSM. Run until the flow field is reasonably converged and then use the `solve/initialize/ init-instantaneous-vel` text command to generate the instantaneous velocity field out of the steady-state RANS results. This command must be executed before LES is enabled. This option is available for all RANS-based models and it will create a much more realistic initial field for the LES run. Additionally, it will help in reducing the time needed for the LES simulation to reach a statistically stable mode. This step is optional.

2. When you enable LES, ANSYS Fluent will automatically turn on the unsteady solver option and choose the second-order implicit formulation. You will need to set the appropriate time step size and all the needed solution parameters. (See User Inputs for Time-Dependent Problems (p. 1609) for guidelines on setting solution parameters for transient calculations in general.) The bounded central differencing spatial discretization scheme will be automatically enabled for momentum equations. Both the bounded central-differencing and pure central-differencing schemes are available for all equations when running LES simulations.

3. Run LES until the flow becomes statistically steady. The best way to see if the flow is fully developed and statistically steady is to monitor forces and solution variables (for example, velocity components or pressure) at selected locations in the flow.

4. Zero out the initial statistics using the `solve/initialize/init-flow-statistics` text command. Before you restart the solution, enable **Data Sampling for Time Statistics** in the Run Calculation Task Page (p. 2537), as described in User Inputs for Time-Dependent Problems (p. 1609). With this option enabled, ANSYS Fluent will gather data for time statistics while performing a large eddy simulation. You can set the **Sampling Interval** such that **Data Sampling for Time Statistics** can be performed at the specified frequency. When **Data Sampling for Time Statistics** is enabled, the statistics collected at each sampling interval can be postprocessed and you can then view both the mean and the root-mean-square-error (RMSE) values in ANSYS Fluent. The **Time Sampled** displays the time period over which data has been sampled for the postprocessing of the mean and RMSE values. As long as the time step size has been constant, dividing this by the time step size yields the number of data sets that have been collected. If the time step size is varied, every contribution of data sets sampled is automatically weighted by the current time step size.

5. Continue until you get statistically stable data. The duration of the simulation can be determined beforehand by estimating the mean flow residence time in the solution domain (L/U, where L is the characteristic length of the solution domain and U is a characteristic mean flow velocity). The simulation should be run for at least a few mean flow residence times.

Instructions for setting the solution parameters for LES are provided below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.19.5.1. Temporal Discretization

ANSYS Fluent provides both first-order and second-order temporal discretizations. For LES, the second-order discretization is recommended.

Solution → **Methods**

11.19.5.2. Spatial Discretization

Overly diffusive schemes such as the first-order upwind or power law scheme should be avoided, because they may unduly damp out the energy of the resolved eddies. The central-differencing based schemes are recommended for all equations when you use the LES model. ANSYS Fluent provides two central-differencing based schemes: *pure* central-differencing and *bounded* central-differencing. The bounded scheme is the default option when you select LES, DES, SDES, or SBES.

Solution → **Methods**

11.20. Postprocessing for Turbulent Flows

ANSYS Fluent provides postprocessing options for displaying, plotting, and reporting various turbulence quantities, which include the main solution variables and other auxiliary quantities.

Turbulence quantities that can be reported for the Spalart-Allmaras model are as follows:

- **Modified Turbulent Viscosity**
- **Turbulent Viscosity**
- **Effective Viscosity**
- **Turbulent Viscosity Ratio**
- **Effective Thermal Conductivity**
- **Effective Prandtl Number**
- **Wall Yplus**
- **Curvature Correction Function fr** (only when the curvature correction is enabled)

Turbulence quantities that can be reported for the DES model in combination with the Spalart-Allmaras model are as follows:

- all of the quantities available for the Spalart-Allmaras model
- **DES TKE Dissipation Multiplier**
- **Normalized Q criterion**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Q criterion**

> **Note**
>
> For a Detached Eddy Simulation with the Spalart-Allmaras model, the **DES TKE Dissipation Multiplier** represents the function f_d in Equation 4.250.

Turbulence quantities that can be reported for the k-ε models are as follows:

- **Turbulent Kinetic Energy (k)**
- **Turbulent Intensity**
- **Turbulent Dissipation Rate (Epsilon)**
- **Production of k**
- **Turbulent Viscosity**
- **Effective Viscosity**
- **Turbulent Viscosity Ratio**
- **Effective Thermal Conductivity**
- **Effective Prandtl Number**
- **Wall Ystar**
- **Wall Yplus**
- **Turbulent Reynolds Number (Re_y)** (only when the enhanced wall treatment is used for the near-wall treatment)
- **Curvature Correction Function fr** (only when the curvature correction is enabled)

Turbulence quantities that can be reported for the DES model in combination with the Realizable k-ε model are as follows:

- all of the quantities available for the Realizable k-ε model
- **DES TKE Dissipation Multiplier**
- **Normalized Q criterion**
- **Q criterion**

> **Note**
>
> For a Detached Eddy Simulation with the Realizable k-ε model, the **DES TKE Dissipation Multiplier** represents the function f_d in Equation 4.257.

Turbulence quantities that can be reported for the k-ω models are as follows:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Turbulent Kinetic Energy (k)**
- **Turbulent Intensity**
- **Turbulent Dissipation Rate (Epsilon)**
- **Intermittency**
- **Specific Dissipation Rate (Omega)**
- **Production of k**
- **Turbulent Viscosity**
- **Effective Viscosity**
- **Turbulent Viscosity Ratio**
- **Effective Thermal Conductivity**
- **Effective Prandtl Number**
- **Wall Ystar**
- **Wall Yplus**
- **Turbulent Reynolds Number (Re_y)**
- **Curvature Correction Function fr** (only when the curvature correction is enabled)

Turbulence quantities that can be reported for the Standard and BSL k-ω model in combination with the SAS model are as follows:

- all of the quantities available for the k-ω model
- **Normalized Q criterion**
- **Q criterion**

Turbulence quantities that can be reported for the BSL or SST k-ω model in combination with the DES model are as follows:

- all of the quantities available for the k-ω models
- **DES TKE Dissipation Multiplier**
- **Normalized Q criterion**
- **Q criterion**

Note

For a Detached Eddy Simulation with BSL / SST, **DES TKE Dissipation Multiplier** represents the function F_{DES} in Equation 4.261 and F_{IDDES} in Equation 4.265, depending on the specified model. Within the DES concept, the **DES TKE Dissipation Multiplier** increases the destruction

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

term in the transport equation for the turbulence kinetic energy in LES regions (F_{DES}>1). This increase in destruction terms reduces the eddy viscosity in LES regions.

Turbulence quantities that can be reported for the BSL or SST k-ω model in combination with the Shielded Detached Eddy Simulation (SDES) model or the Stress-Blended Eddy Simulation (SBES) model are as follows:

- all of the quantities available for the k-ω models
- **Shielding Function for SBES or SDES**
- **Normalized Q criterion**
- **Q criterion**

Turbulence quantities that can be reported for the transition k-kl-ω model are as follows:

- **Turbulent Kinetic Energy (k)**
- **Laminar Kinetic Energy**
- **Total Fluctuation Energy**
- **Turbulent Intensity**
- **Turbulent Dissipation Rate (Epsilon)**
- **Specific Dissipation Rate (Omega)**
- **Production of k**
- **Production of laminar k**
- **Turbulent Viscosity**
- **Turbulent Viscosity (large-scale)**
- **Turbulent Viscosity (small-scale)**
- **Effective Viscosity**
- **Turbulent Viscosity Ratio**
- **Effective Thermal Conductivity**
- **Effective Prandtl Number**
- **Wall Ystar**
- **Wall Yplus**
- **Turbulent Reynolds Number (Re_y)**

Turbulence quantities that can be reported for the Transition SST model are as follows:

- **Turbulent Kinetic Energy (k)**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Turbulent Intensity**
- **Turbulent Dissipation Rate (Epsilon)**
- **Intermittency**
- **Intermittency Effective**
- **Momentum Thickness Re**
- **Geometric Roughness Height**
- **Specific Dissipation Rate (Omega)**
- **Production of k**
- **Turbulent Viscosity**
- **Effective Viscosity**
- **Turbulent Viscosity Ratio**
- **Effective Thermal Conductivity**
- **Effective Prandtl Number**
- **Wall Ystar**
- **Wall Yplus**
- **Turbulent Reynolds Number (Re_y)**
- **Curvature Correction Function fr** (only when the curvature correction is enabled)

Turbulence quantities that can be reported for the Transition SST model in combination with the SAS model are as follows:

- all of the quantities available for the Transition SST model
- **Normalized Q criterion**
- **Q criterion**

Turbulence quantities that can be reported for the Transition SST model in combination with the DES model are as follows:

- all of the quantities available for the Transition SST model
- **DES TKE Dissipation Multiplier**
- **Normalized Q criterion**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Q criterion**

Note

For the Transition SST model in combination with the DES model, **DES TKE Dissipation Multiplier** represents the function F_{DES} in Equation 4.264 and F_{IDDES} in Equation 4.265 depending on the specified model. Within the DES concept, the **DES TKE Dissipation Multiplier** increases the destruction term in the transport equation for the turbulence kinetic energy in LES regions (F_{DES}>1). This increase in destruction terms reduces the eddy viscosity in LES regions.

Turbulence quantities that can be reported for the Transition SST model in combination with the Shielded Detached Eddy Simulation (SDES) model or the Stress-Blended Eddy Simulation (SBES) model are as follows:

- all of the quantities available for the Transition SST model
- **Shielding Function for SBES or SDES**
- **Normalized Q criterion**
- **Q criterion**

Turbulence quantities that can be reported for the RSM are as follows:

- **Turbulent Kinetic Energy (k)**
- **Turbulent Intensity**
- **UU Reynolds Stress**
- **VV Reynolds Stress**
- **WW Reynolds Stress**
- **UV Reynolds Stress**
- **VW Reynolds Stress**
- **UW Reynolds Stress**
- **Turbulent Dissipation Rate (Epsilon)**
- **Specific Dissipation Rate (Omega)** (ω-based Reynolds stress models only)
- **Production of k**
- **Turbulent Viscosity**
- **Effective Viscosity**
- **Turbulent Viscosity Ratio**
- **Effective Thermal Conductivity**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Effective Prandtl Number**
- **Wall Ystar**
- **Wall Yplus**
- **Turbulent Reynolds Number (Re_y)** (ε-based Reynolds stress models only)

Turbulence quantities that can be reported for the ω-based Reynolds stress models in combination with the SAS model are as follows:

- all of the quantities available for the RSM model
- **Normalized Q criterion**
- **Q criterion**

Turbulence quantities that can be reported for the SAS model in combination with the SST k-ω model are as follows:

- **Turbulent Kinetic Energy (k)**
- **Turbulent Intensity**
- **Turbulent Dissipation Rate (Epsilon)**
- **Intermittency**
- **Specific Dissipation Rate (Omega)**
- **Production of k**
- **Turbulent Viscosity**
- **Effective Viscosity**
- **Turbulent Viscosity Ratio**
- **Effective Thermal Conductivity**
- **Effective Prandtl Number**
- **Wall Yplus**
- **Wall Ystar**
- **Curvature Correction Function fr** (only when the curvature correction is enabled)
- **Normalized Q criterion**
- **Q criterion**

Turbulence quantities that can be reported for the LES model are as follows:

- **Turbulence Kinetic Energy**
- **Turbulence Intensity**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Subgrid Kinetic Energy**
- **Production of k**
- **Subgrid Turbulent Viscosity**
- **Subgrid Effective Viscosity**
- **Subgrid Turbulent Viscosity Ratio**
- **Subgrid Filter Length**
- **Subgrid Test-Filter Length**
- **Subgrid Dissipation Rate**
- **Subgrid Dynamic Viscosity Const**
- **Subgrid Dynamic Prandtl Number**
- **Subgrid Dynamic Sc of Species**
- **Subtest Kinetic Energy**
- **Effective Thermal Conductivity**
- **Effective Prandtl Number**
- **Wall Ystar**
- **Wall Yplus**
- **Normalized Q criterion**
- **Q criterion**

Additional turbulence quantities can be reported for the Embedded LES (ELES) model.

Note

Within the Embedded LES zone, the modeled eddy viscosity is determined by an algebraic subgrid-scale eddy viscosity model. The global turbulence model is either frozen or, if it is either SAS, DES, SDES, or SBES with an underlying two-equation RANS model, runs in a "passive mode", that is without affecting the momentum equations.

Hence, most turbulence postprocessing in the Embedded LES zone refers to the frozen or "passive" global turbulence model. Special care is necessary with the modeled eddy viscosity, because there are two to consider: one from the frozen / passive global turbulence model; and one from the "local" algebraic subgrid-scale model that is running within the Embedded LES zone and actually affects the momentum equations.

- Some postprocessing quantities refer to a "passive" global turbulence model or are zero if the global model cannot run in "passive" mode and therefore is frozen in the Embedded LES zone.
  - **Turbulent Viscosity**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **Turbulent Viscosity Ratio**

- In the Embedded LES zone, the subgrid-scale eddy viscosity from the local algebraic model, which actually affects the momentum equations, is displayed as:

  - **LES Subgrid Turbulent Viscosity**

- The following quantities are specific to the Dynamic LES subgrid-scale eddy viscosity models. They are available in Embedded LES zones if the **Dynamic Smagorinsky** model is used in any Embedded LES zone; they are also available for global LES when the **Smagorinsky-Lilly** subgrid-scale model is selected with the **Dynamic Stress** option enabled or when the **Kinetic-Energy Transport** subgrid-scale model is selected:

  - **Subgrid Test-Filter Length**

  - **Subgrid Dynamic Viscosity Const**

  - **Subtest Kinetic Energy**

All of these variables can be found in the **Turbulence...** category of the variable selection drop-down list that appears in postprocessing dialog boxes. See Field Function Definitions (p. 1931) for their definitions.

For additional information, see the following sections:
11.20.1. Custom Field Functions for Turbulence
11.20.2. Postprocessing Turbulent Flow Statistics
11.20.3. Troubleshooting

11.20.1. Custom Field Functions for Turbulence

In addition to the quantities listed in Postprocessing for Turbulent Flows (p. 814), above, you can define your own turbulence quantities using the Custom Field Function Calculator Dialog Box (p. 2683).

User-Defineed → Field Functions → Custom...

The following functions may be useful:

- the ratio of production of k to its dissipation ($G_k / \rho\varepsilon$)

- the ratio of the mean flow to turbulent time scale, η ($\equiv Sk / \varepsilon$)

- the Reynolds stresses derived from the Boussinesq formula (for example, $-\overline{uv} = \nu_t \frac{\partial u}{\partial y}$)

11.20.2. Postprocessing Turbulent Flow Statistics

As described in Large Eddy Simulation (LES) Model (in the Theory Guide), LES involves the solution of a transient flow field, but it is the mean flow quantities that are of interest from an engineering standpoint.

For all other turbulent flow, if **Data Sampling for Time Statistics** is enabled in the Run Calculation Task Page (p. 2537), ANSYS Fluent gathers data for time statistics while performing the simulation. The statistics that ANSYS Fluent collects at each sampling interval (which consists of the mean and the root-mean-square-error (RMSE) values) can be postprocessed by selecting **Unsteady Statistics...** in any of the postprocessing dialog boxes. You can view several variables that include, but are not limited to, shear stresses (**Resolved UV/UW/VW Reynolds Stress**), flow heat fluxes (**Resolved UT/VT/WT Heat**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Flux), and species statistics (**RMSE Mass Fraction of** species and **Mean Mass Fraction** of species). If you select **Unsteady Wall Statistics...** in any of the postprocessing dialog boxes, you can view wall statistics such as **Mean Pressure Coefficient**, **Mean Wall Shear Stress**, **Mean X-Wall Shear Stress**, **Mean Y-Wall Shear Stress**, **Mean Z-Wall Shear Stress**, **Mean Skin Friction Coefficient**, **Mean Surface Heat Flux**, **Mean Surface Heat Transfer Coef.**, **Mean Surface Nusselt Number**, **Mean Surface Stanton Number**. Note that these quantities are only the statistical evaluation of sampled solution data, and do not contain any kind of modeled turbulent fluctuations. See Postprocessing for Time-Dependent Problems (p. 1622) for details.

The **Time Sampled** displays the time period over which data has been sampled for the postprocessing of the mean and RMSE values. As long as the time step size has been constant, dividing this by the time step size yields the number of data sets that have been collected. If the time step size is varied, every contribution of data sets sampled is automatically weighted by the current time step size.

Important

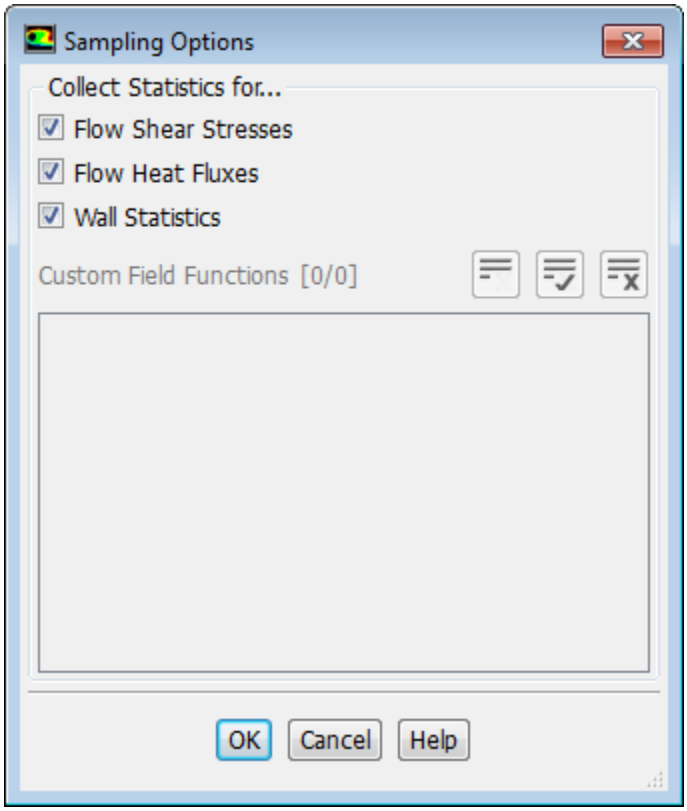
Note that *mean* statistics are collected only in interior cells and not on wall surfaces. Therefore, when node or cell values of mean quantities are plotted on the wall surface, you are actually plotting values in nearby cells attached to the wall.

If you want to control what set of variables are available for postprocessing, open the **Sampling Options** dialog box and enable or disable the statistics shown in Figure 11.26: The Sampling Options Dialog Box (p. 824).

Solution → **Run Calculation** → **Sampling Options...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.26: The Sampling Options Dialog Box



Important

When including or excluding statistics on variables, it is recommended that you re-initialize the flow statistics.

11.20.3. Troubleshooting

You can use the postprocessing options not only for the purpose of interpreting your results but also for investigating any anomalies that may appear in the solution. For instance, you may want to plot contours of the k field to check if there are any regions where k is erroneously large or small. You should see a high k region in the region where the production of k is large. You may want to display the turbulent viscosity ratio field in order to see whether or not the turbulence takes full effect. Usually the turbulent viscosity is at least two orders of magnitude larger than molecular viscosity for fully-developed turbulent flows modeled using the RANS approach (that is, not using LES). You may also want to check whether you are using an adequate near-wall mesh when using a y^+-insensitive wall treatment (see Grid Resolution for RANS Models (p. 756) for guidelines). For EWT-ε, you can also display filled contours of Re_y (turbulent Reynolds number) overlaid on the mesh (see Enhanced Wall Treatment ε-Equation (EWT-ε) in the Theory Guide for more details).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 12: Modeling Heat Transfer

This chapter provides details about the heat transfer models available in ANSYS Fluent. Information is presented in the following sections:

12.1. Introduction
12.2. Modeling Conductive and Convective Heat Transfer
12.3. Modeling Radiation
12.4. Modeling Periodic Heat Transfer

12.1. Introduction

The flow of thermal energy from matter occupying one region in space to matter occupying a different region in space is known as heat transfer. Heat transfer can occur by three main mechanisms: conduction, convection, and radiation. Physical models involving conduction and/or convection only are the simplest (Modeling Conductive and Convective Heat Transfer (p. 825)), while buoyancy-driven flow or natural convection (Natural Convection and Buoyancy-Driven Flows (p. 833)), and flow involving radiation (Modeling Radiation (p. 845)) are more complex. Depending on your problem, ANSYS Fluent will solve a variation of the energy equation that takes into account the heat transfer methods you have specified. ANSYS Fluent is also able to predict heat transfer in periodically repeating geometries (Modeling Periodic Heat Transfer (p. 912)), therefore greatly reducing the required computational effort in certain cases.

12.2. Modeling Conductive and Convective Heat Transfer

ANSYS Fluent allows you to include heat transfer within the fluid and/or solid regions in your model. Problems ranging from thermal mixing within a fluid to conduction in composite solids can therefore be handled by ANSYS Fluent.

When your ANSYS Fluent model includes heat transfer you will need to enable the relevant physical models, supply thermal boundary conditions, and enter material properties (which may vary with temperature) that govern heat transfer. For information about heat transfer theory, see Heat Transfer Theory in the Theory Guide. Information about heat transfer theory and how to set up and use heat transfer in your ANSYS Fluent model is presented in the following subsections:

12.2.1. Solving Heat Transfer Problems
12.2.2. Solution Strategies for Heat Transfer Modeling
12.2.3. Postprocessing Heat Transfer Quantities
12.2.4. Natural Convection and Buoyancy-Driven Flows
12.2.5. Shell Conduction Considerations

12.2.1. Solving Heat Transfer Problems

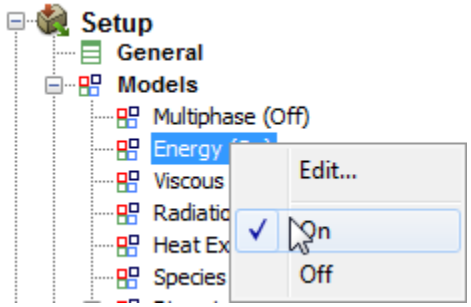
The procedure for setting up a heat transfer problem is described below. Note that this procedure includes only those steps necessary for the heat transfer model itself; you will need to define the other settings (for example, other models, boundary conditions) as usual.

1. To activate the calculation of heat transfer, enable the energy equation by right-clicking **Energy** in the tree (under **Setup/Models**) and clicking **On** in the menu that opens.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → Models → Energy → On

Figure 12.1: Enabling the Energy Equation



2. (Optional, pressure-based solver only.) If you are modeling viscous flow and you want to include the viscous heating terms in the energy equation, enable the **Viscous Heating** option in the **Viscous Model** dialog box.

   Setup → Models → Viscous → Edit...

   As noted in Inclusion of the Viscous Dissipation Terms in the Theory Guide, the viscous heating terms in the energy equation are (by default) ignored by ANSYS Fluent when the pressure-based solver is used. They are always included for the density-based solver. Viscous dissipation should be enabled when the shear stress in the fluid is large (for example, in lubrication problems) and/or in high-velocity, compressible flows. See Equation 5.9 in the Theory Guide.

3. Define thermal boundary conditions at flow inlets, flow outlets, and walls. The boundary condition dialog boxes can be opened by right-clicking the boundary name in the tree (under **Setup/Boundary Conditions**) and clicking **Edit...** in the menu that opens; alternatively, you can open them from the **Boundary Conditions** task page:

   Setup → Boundary Conditions

   At flow inlets and exits you will set the temperature; at walls you may use any of the following thermal conditions:

   - specified heat flux
   - specified temperature
   - convective heat transfer
   - external radiation
   - combined external radiation and external convective heat transfer

   Thermal Boundary Conditions at Walls (p. 370) provides details on the model inputs that govern these thermal boundary conditions. The default thermal boundary condition at inlets is a specified

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

temperature of 300 K; at walls the default condition is zero heat flux (adiabatic). See Cell Zone and Boundary Conditions (p. 237) for details about boundary condition inputs.

Important

If your heat transfer application involves two separated fluid regions, see the information provided below.

4. Define material properties for heat transfer.

   Setup → Materials

   Heat capacity and thermal conductivity must be defined, and you can specify many properties as functions of temperature as described in Physical Properties (p. 459).

12.2.1.1. Limiting the Predicted Temperature Range

For stability reasons, ANSYS Fluent includes a limit on the predicted temperature range. The purpose of the temperature ceiling and floor is to improve the stability of calculations in which the temperature should physically lie within known limits. Sometimes intermediate solutions of the equations give rise to temperatures beyond these limits for which property definitions, and so on, are not well defined.

The temperature limits keep the temperatures within the expected range for your problem. If the ANSYS Fluent calculation predicts a temperature above the maximum limit, the stored temperature values are "pegged" at this maximum value. The default for the temperature ceiling is 5000 K. If the ANSYS Fluent calculation predicts a temperature below the minimum limit, the stored temperature values are "pegged" at this minimum value. The default for the temperature minimum is 1 K.

If you expect the temperature in your domain to exceed 5000 K, use the **Solution Limits** dialog box to increase the **Maximum Static Temperature**.

Solution → Controls → Limits...

12.2.1.2. Modeling Heat Transfer in Two Separated Fluid Regions

If your heat transfer application involves two fluid regions separated by a solid zone or a wall, as illustrated in Figure 12.2: Typical Counterflow Heat Exchanger Involving Heat Transfer Between Two Separated Fluid Streams (p. 828), you will need to define the problem with some care. Specifically:

- You should not use outflow boundary conditions in either fluid.
- You can establish separate fluid properties by selecting a different fluid material for each zone. For species calculations, however, you can only select a single mixture material for the entire domain.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 12.2: Typical Counterflow Heat Exchanger Involving Heat Transfer Between Two Separated Fluid Streams

12.2.2. Solution Strategies for Heat Transfer Modeling

Although many simple heat transfer problems can be successfully solved using the default solution parameters assumed by ANSYS Fluent, you may accelerate the convergence of your problem and/or improve the stability of the solution process using some of the guidelines provided in this section.

12.2.2.1. Under-Relaxation of the Energy Equation

When you use the pressure-based solver, ANSYS Fluent under-relaxes the energy equation using the under-relaxation parameter defined by you in the **Solution Controls** task page, as described in Setting Under-Relaxation Factors (p. 1564).

Solution → **Controls**

If you are using the non-adiabatic non-premixed combustion model, you will set the energy under-relaxation factor as usual but you will also set an under-relaxation factor for temperature, as described below.

ANSYS Fluent uses a default under-relaxation factor of 1.0 for the energy equation, regardless of the form in which it is solved (temperature or enthalpy). In problems where the energy field impacts the fluid flow (via temperature-dependent properties or buoyancy) you should use a lower value for the under-relaxation factor, in the range of 0.8–1.0. In problems where the flow field is decoupled from the temperature field (no temperature-dependent properties or buoyancy forces), you can usually retain the default value of 1.0.

12.2.2.2. Under-Relaxation of Temperature When the Enthalpy Equation is Solved

When the enthalpy form of the energy equation is solved (that is, when you are using the non-adiabatic non-premixed combustion model), ANSYS Fluent also under-relaxes the temperature, updating the temperature by only a fraction of the change that would result from the change in the (under-relaxed) enthalpy values. This second level of under-relaxation can be used to good advantage when you would like to let the enthalpy field change rapidly, but the temperature response (and its effect on fluid properties) to lag. ANSYS Fluent uses a default setting of 1.0 for the under-relaxation on temperature and you can modify this setting using the **Solution Controls** task page.

Solution → **Controls**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12.2.2.3. Disabling the Species Diffusion Term

If you are solving for species transport using the pressure-based solver and you encounter convergence difficulties, you may want to consider disabling the **Diffusion Energy Source** option in the **Species Model** dialog box.

Setup → Models → Species Edit...

When this option is disabled, ANSYS Fluent will neglect the effects of species diffusion on the energy equation.

Note that species diffusion effects are always included when the density-based solver is used.

12.2.2.4. Step-by-Step Solutions

Often the most efficient strategy for predicting heat transfer is to compute an isothermal flow first and then add the calculation of the energy equation. The procedure differs slightly, depending on whether or not the flow and heat transfer are coupled.

12.2.2.4.1. Decoupled Flow and Heat Transfer Calculations

If your flow and heat transfer are decoupled (no temperature-dependent properties or buoyancy forces), you can first solve the isothermal flow (energy equation turned off) to yield a converged flow-field solution and then solve the energy transport equation alone.

Important

Since the density-based solver always solves the flow and energy equations together, the procedure for solving for energy alone applies to the pressure-based solver, only.

You can temporarily disable the flow equations or the energy equation by deselecting them in the **Equations** dialog box:

Solution → Controls Equations...

You can also disable the energy equation:

Setup → Models → Energy Off

12.2.2.4.2. Coupled Flow and Heat Transfer Calculations

If the flow and heat transfer are coupled (that is, your model includes temperature-dependent properties or buoyancy forces), you can first solve the flow equations before enabling energy. Once you have a converged flow-field solution, you can enable energy and solve the flow and energy equations simultaneously to complete the heat transfer simulation.

12.2.2.5. Specifying a Solid Time step

For transient conjugate heat transfer (CHT) problems, particularly those with combustion, the dominant time-scales in the fluid and solid zones are often quite different. In most cases, it is desirable to have a

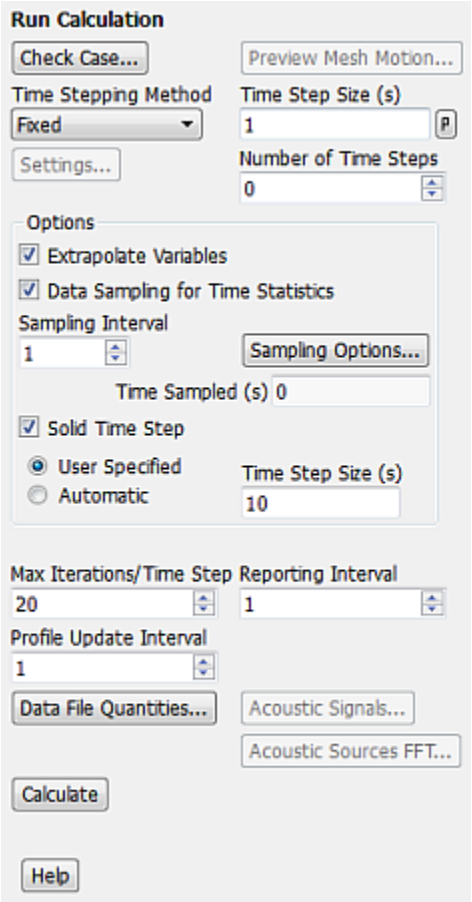
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

larger time step in solid zones, while maintaining a smaller time step in fluid zones. This will increase the speed at which the solid heat transfer reaches steady-state without compromising the solution accuracy of the fluid flow.

You can specify a solid time step on the **Run Calculation** task page:

Solution → Run Calculation

Figure 12.3: The Run Calculation Task Page Showing Solid Time Step



1. Under **Options**, select **Solid Time Step**.
2. Use the default **Automatic** time step or select **User Specified** to enter your own time step.

12.2.2.5.1. Automatic Time Step Calculation

The default **Automatic** time step is calculated by ANSYS Fluent using the following formula:

$$\Delta t_{solid} = \frac{L_{scale_solid}^2}{\alpha} \tag{12.1}$$

where,

- L_{scale_solid} is the representative length scale and is calculated as $\sqrt[3]{Solid_Volume}$.
- α is $\frac{k}{\rho c_p}$, the representative velocity scale, where k is the conductivity, ρ is the density, and c_p is the specific heat capacity of the solid.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The calculation for Δt_{solid} results in an approximation for the solid time step which might aide in the solution running more efficiently. It should be noted, however, that this is only an engineering approximation as there is no general way to calculate a meaningful length scale for an arbitrary geometry.

12.2.3. Postprocessing Heat Transfer Quantities

For information about postprocessing heat transfer quantities, see the following sections:

12.2.3.1. Available Variables for Postprocessing
12.2.3.2. Definition of Enthalpy and Energy in Reports and Displays
12.2.3.3. Reporting Heat Transfer Through Boundaries
12.2.3.4. Reporting Heat Transfer Through a Surface
12.2.3.5. Reporting Averaged Heat Transfer Coefficients
12.2.3.6. Exporting Heat Flux Data

12.2.3.1. Available Variables for Postprocessing

ANSYS Fluent provides reporting options for simulations involving heat transfer. You can generate graphical plots or reports of the following variables/functions:

- **Static Temperature**
- **Total Temperature**
- **Enthalpy**
- **Relative Total Temperature**
- **Rothalpy**
- **Wall Temperature**
- **Wall Temperature (Thin)**
- **Total Enthalpy**
- **Total Enthalpy Deviation**
- **Entropy**
- **Total Energy**
- **Internal Energy**
- **Total Surface Heat Flux**
- **Surface Heat Transfer Coef.**
- **Surface Nusselt Number**
- **Surface Stanton Number**

The first 12 variables listed above are contained in the **Temperature...** category of the variable selection drop-down list that appears in postprocessing dialog boxes, and the remaining variables are in the **Wall Fluxes...** category. See Field Function Definitions (p. 1931) for their definitions.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12.2.3.2. Definition of Enthalpy and Energy in Reports and Displays

The reported enthalpy values may be different between the pressure-based and density-based solver depending on the properties of the working fluid, compressible, incompressible, ideal gas, etc. For details, see the discussion of enthalpy in The Energy Equation in chapter 5, Heat Transfer of the Theory Guide.

12.2.3.3. Reporting Heat Transfer Through Boundaries

You can use the **Flux Reports** dialog box to compute the heat transfer through each boundary of the domain, or to sum the heat transfer through all boundaries to check the heat balance.

Results → **Reports** → **Fluxes** **Edit...**

It is recommended that you perform a heat balance check to ensure that your solution is truly converged.

12.2.3.4. Reporting Heat Transfer Through a Surface

You can use the **Surface Integrals** dialog box to compute the heat transfer through any boundary or any surface created using the methods described in Creating Surfaces and Cell Registers for Displaying and Reporting Data (p. 1709).

Results → **Reports** → **Surface Integrals** **Edit...**

To report the mass flow rate of enthalpy

$$Q = \int H \rho \vec{v} \cdot d\vec{A} \tag{12.2}$$

choose **Flow Rate** for the **Report Type** in the **Surface Integrals** dialog box, select **Enthalpy** (in the **Temperature...** category) as the **Field Variable**, and select the surface(s) on which to integrate.

12.2.3.5. Reporting Averaged Heat Transfer Coefficients

The **Surface Integrals** dialog box can also be used to generate a report of averaged heat transfer coefficient h on a surface ($\frac{1}{A}\int h dA$).

Results → **Reports** → **Surface Integrals** **Edit...**

In the **Surface Integrals** dialog box, choose **Area-Weighted Average** for **Report Type**, select **Surface Heat Transfer Coef.** (in the **Wall Fluxes...** category) as the **Field Variable**, and select the surface.

12.2.3.6. Exporting Heat Flux Data

It is possible to export heat flux data on wall zones (including radiation) to a generic file that you can examine or use in an external program. To save a heat flux file, you will use the `custom-heat-flux` text command.

`file` → `export` → `custom-heat-flux`

Heat transfer data will be exported in the following free format for each face zone that you select for export:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
zone-name nfaces
x_f y_f z_f A Q T_w T_c HTC
.
.
.
```

Each block of data starts with the name of the face zone (`zone-name`) and the number of faces in the zone (`nfaces`). Next there is a line for each face (that is, `nfaces` lines), each containing the components of the face centroid (`x_f`, `y_f`, and, in 3D, `z_f`), the face area (A), the heat transfer rate (Q), the face temperature (T_w), the adjacent cell temperature (T_c), and the heat transfer coefficient (HTC). If the heat transfer coefficient is calculated based on wall function (Equation 33.55 (p. 2001)), then Q is the convective heat transfer rate. Otherwise, Q will be the total heat transfer rate, including radiation heat transfer.

12.2.4. Natural Convection and Buoyancy-Driven Flows

When heat is added to a fluid and the fluid density varies with temperature, a flow can be induced due to the force of gravity acting on the density variations. Such buoyancy-driven flows are termed natural-convection (or mixed-convection) flows and can be modeled by ANSYS Fluent.

For more information about the theory behind natural convection and buoyancy-driven flows, see Natural Convection and Buoyancy-Driven Flows Theory in the Theory Guide.

12.2.4.1. Modeling Natural Convection in a Closed Domain

When you model natural convection inside a closed domain, the solution will depend on the mass inside the domain. Since this mass will not be known unless the density is known, you must model the flow in one of the following ways:

- Perform a transient calculation. In this approach, the initial density will be computed from the initial pressure and temperature, so the initial mass is known. As the solution progresses over time, this mass will be properly conserved. If the temperature differences in your domain are large, you must follow this approach.
- Perform a steady-state calculation using the Boussinesq model (described in The Boussinesq Model (p. 833)). In this approach, you will specify a constant density, so the mass is properly specified. This approach is valid only if the temperature differences in the domain are small. If not, you use the transient approach.

Important

For a closed domain, you can use the *incompressible* ideal gas law only with a *fixed* operating pressure. It *cannot* be used with a floating operating pressure. You can use the *compressible* ideal gas law with either *floating* or *fixed* operating pressure.

See Floating Operating Pressure (p. 597) for more information about the floating operating pressure option.

12.2.4.2. The Boussinesq Model

For many natural-convection flows, you can get faster convergence with the Boussinesq model than you can get by setting up the problem with fluid density as a function of temperature. This model treats density as a constant value in all solved equations, except for the buoyancy term in the momentum equation:

$$\left(\rho-\rho_0\right)g \approx -\rho_0\beta\left(T-T_0\right)g \tag{12.3}$$

where ρ_0 is the (constant) density of the flow, T_0 is the operating temperature, and β is the thermal expansion coefficient. Equation 12.3 (p. 833) is obtained by using the Boussinesq approximation $\rho=\rho_0\left(1-\beta\Delta T\right)$ to eliminate ρ from the buoyancy term. This approximation is accurate as long as changes in actual density are small; specifically, the Boussinesq approximation is valid when $\beta\left(T-T_0\right)\ll 1$.

12.2.4.3. Limitations of the Boussinesq Model

The Boussinesq model should not be used if the temperature differences in the domain are large. In addition, it cannot be used with species calculations, combustion, or reacting flows.

12.2.4.4. Steps in Solving Buoyancy-Driven Flow Problems

The procedure for including buoyancy forces in the simulation of mixed or natural convection flows is described below.

1. To activate the calculation of heat transfer, enable the energy equation by right-clicking **Energy** in the tree (under **Setup/Models**) and clicking **On** in the menu that opens (Figure 12.1: Enabling the Energy Equation (p. 826)).

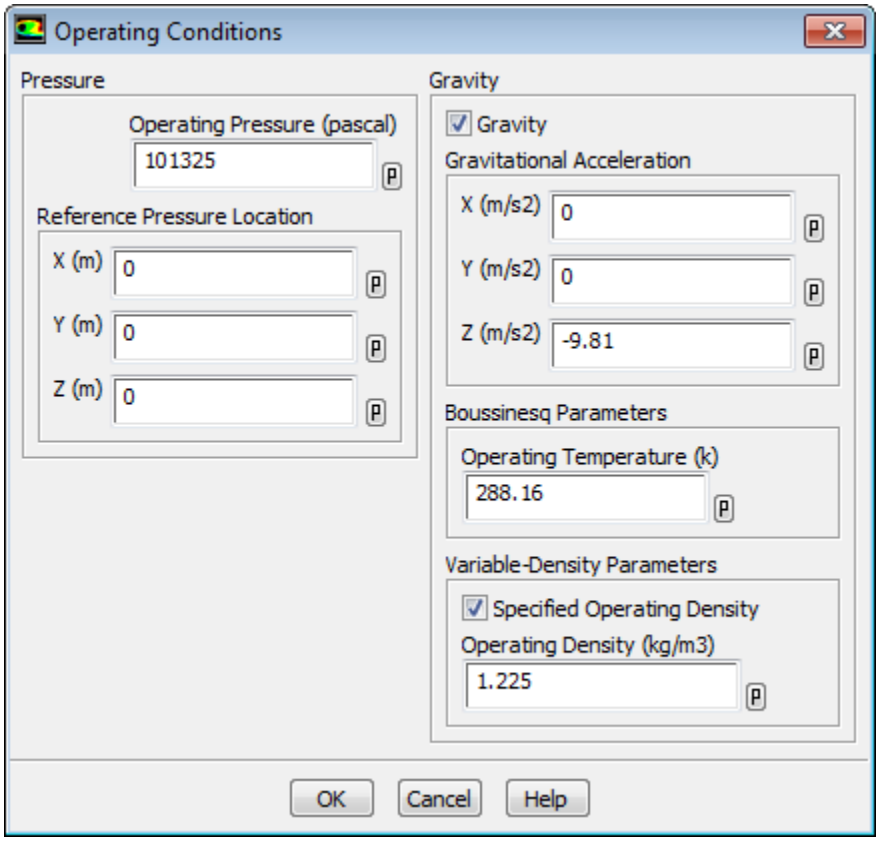
   Setup → **Models** → **Energy** → **On**

2. Define the operating conditions in the **Operating Conditions** dialog box (Figure 12.4: The Operating Conditions Dialog Box (p. 835)).

   Setup → **Cell Zone Conditions** → **Operating Conditions...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 12.4: The Operating Conditions Dialog Box



a. Enable the **Gravity** option under **Gravity**.

b. Enter the appropriate values in the **X**, **Y**, and (for 3D) **Z** fields for **Gravitational Acceleration** for each Cartesian coordinate direction. (Note that the default gravitational acceleration in ANSYS Fluent is zero.)

c. If you are using the incompressible ideal gas law, check that the **Operating Pressure** is set to an appropriate (nonzero) value.

d. Depending on whether or not you use the Boussinesq approximation, specify the appropriate parameters described below:

- If you are not using the Boussinesq model, the inputs are as follows:

  i. If necessary, enable the **Specified Operating Density** option in the **Operating Conditions** dialog box, and enter a value for the **Operating Density**. See below for details.

  ii. Define the fluid density as a function of temperature as described in Defining Properties Using Temperature-Dependent Functions (p. 475) and Density (p. 479).

  Setup → Materials

- If you are using the Boussinesq model (described in The Boussinesq Model (p. 833)) the inputs are as follows:

  i. Enter the **Operating Temperature** (T_0 in Equation 12.3 (p. 833)) in the **Operating Conditions** dialog box.

ii. Select **boussinesq** in the drop-down list for **Density** in the **Create/Edit Materials** dialog box as described in Defining Properties Using Temperature-Dependent Functions (p. 475) and Density (p. 479), and enter a constant value.

iii. Also in the **Create/Edit Materials** dialog box, enter an appropriate value for the **Thermal Expansion Coefficient** (β in Equation 12.3 (p. 833)) for the fluid material.

Note that if your model involves multiple fluid materials you can choose whether or not to use the Boussinesq model for each material. As a result, you may have some materials using the Boussinesq model and others not. In such cases, you will need to set all the parameters described above in this step.

3. Define the boundary conditions. The boundary condition dialog boxes can be opened by right-clicking the boundary name in the tree (under **Setup/Boundary Conditions**) and clicking **Edit...** in the menu that opens; alternatively, you can open them from the **Boundary Conditions** task page:

Setup → **Boundary Conditions**

The boundary pressures that you specify at pressure inlet and outlet boundaries are the redefined pressures as given by Equation 12.4 (p. 836). In general you should enter equal pressures, p', at the inlet and exit boundaries of your ANSYS Fluent model if there are no externally imposed pressure gradients.

4. Set the parameters that control the solution, using the **Solution Methods** task page.

Solution → **Methods**

a. Select **Body Force Weighted** or **Second Order** in the drop-down list for **Pressure** under **Spatial Discretization** in the **Solution Methods** task page.

b. If you are using the pressure-based solver, selecting **PRESTO!** for **Pressure** under **Spatial Discretization** is recommended.

c. Add cells near the walls to resolve boundary layers, if necessary.

See also Solving Heat Transfer Problems (p. 825) for information on setting up heat transfer calculations.

12.2.4.5. Operating Density

When the Boussinesq approximation is not used, the operating density ρ_0 appears in the body-force term in the momentum equations as $\left(\rho-\rho_0\right)g$. This form of the body-force term follows from the redefinition of pressure in ANSYS Fluent as

$$p_s'=p_s-\rho_0 g x \tag{12.4}$$

The hydrostatic pressure in a fluid at rest is then

$$p_s'=0 \tag{12.5}$$

12.2.4.5.1. Setting the Operating Density

By default, ANSYS Fluent will compute the operating density by averaging over all cells. In some cases, you may obtain better results if you explicitly specify the operating density instead of having the solver compute it for you. For example, if you are solving a natural-convection problem with a pressure

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

boundary, it is important to understand that the pressure you are specifying is p_s' in Equation 12.4 (p. 836). Although you will know the actual pressure p_s, you will need to know the operating density ρ_0 in order to determine p_s' from p_s. Therefore, you should explicitly specify the operating density rather than use the computed average. The specified value should, however, be representative of the average value.

In some cases the specification of an operating density will improve convergence behavior, rather than the actual results. For such cases use the approximate bulk density value as the operating density and be sure that the value you choose is appropriate for the characteristic temperature in the domain.

Note that if you are using the Boussinesq approximation for all fluid materials, the operating density ρ_0 does not appear in the body-force term of the momentum equation. Consequently, you need not specify it.

12.2.4.6. Solution Strategies for Buoyancy-Driven Flows

For high-Rayleigh-number flows you may want to consider the solution guidelines below. In addition, the guidelines presented in Solution Strategies for Heat Transfer Modeling (p. 828) for solving other heat transfer problems can also be applied to buoyancy-driven flows. No steady-state solution exists for some laminar, high-Rayleigh-number flows.

12.2.4.6.1. Guidelines for Solving High-Rayleigh-Number Flows

When you are solving a high-Rayleigh-number flow ($Ra > 10^8$), you should follow one of the procedures outlined below for best results.

The first procedure uses a steady-state approach:

1. Start the solution with a lower value of Rayleigh number (for example, 10^7) and run it to convergence using the first-order scheme.
2. To change the effective Rayleigh number, change the value of gravitational acceleration (for example, from 9.8 to 0.098 to reduce the Rayleigh number by two orders of magnitude).
3. Use the resulting data file as an initial guess for the higher Rayleigh number and start the higher-Rayleigh-number solution using the first-order scheme.
4. After you obtain a solution with the first-order scheme you may continue the calculation with a higher-order scheme.

The second procedure uses a time-dependent approach to obtain a steady-state solution [41] (p. 2889):

1. Start the solution from a steady-state solution obtained for the same or a lower Rayleigh number.
2. Estimate the time constant as [10] (p. 2887)

$$\tau = \frac{L}{U} \sim \frac{L^2}{\alpha}\left(PrRa\right)^{-1/2} = \frac{L}{\sqrt{g\beta\Delta TL}} \tag{12.6}$$

   where L and U are the length and velocity scales, respectively. Use a time step Δt such that

$$\Delta t \approx \frac{\tau}{4} \tag{12.7}$$

   Using a larger time step Δt may lead to divergence.

3. After oscillations with a typical frequency of $f\tau$= 0.05–0.09 have decayed, the solution reaches steady-state. Note that τ is the time constant estimated in Equation 12.6 (p. 837) and f is the oscillation frequency in Hz. In general this solution process may take as many as 5000 time steps to reach steady-state.

12.2.4.7. Postprocessing Buoyancy-Driven Flows

The postprocessing reports of interest for buoyancy-driven flows are the same as for other heat transfer calculations. See Postprocessing Heat Transfer Quantities (p. 831) for details.

12.2.5. Shell Conduction Considerations

For information about shell conduction considerations, see the following sections:

12.2.5.1. Introduction
12.2.5.2. Physical Treatment
12.2.5.3. Limitations of Shell Conduction Walls
12.2.5.4. Managing Shell Conduction Walls
12.2.5.5. Initializing Shells
12.2.5.6. Locking the Temperature for Shells
12.2.5.7. Postprocessing Shells

12.2.5.1. Introduction

By default, ANSYS Fluent treats walls as having zero-thickness and presenting no thermal resistance to heat transfer across them. If a thickness is specified for a wall (thereby making it a thin wall, as described in Thin-Wall Thermal Resistance Parameters (p. 373)) then the appropriate thermal resistance across the wall thickness is imposed, although conduction is considered in the wall in the normal direction only. There are applications, however, where conduction in the planar directions of the wall is also important. For these applications, you have two options: you can either mesh the thickness, or you can use the shell conduction approach. Shell conduction can be used to model one or more layers of wall cells without the need to mesh the wall thickness in a preprocessor. When the shell conduction approach is utilized, you have the ability to easily switch on and off conjugate heat transfer on any wall. If you create a shell—either by enabling **Shell Conduction** in an individual **Wall** dialog box (as described in Shell Conduction (p. 376)) or by defining multiple walls as shell conduction zones using the **Shell Conduction Manager** dialog box (as described in Managing Shell Conduction Walls (p. 840))—and define the settings for the shell layer(s) using the **Shell Conduction Layers** dialog box, then ANSYS Fluent will automatically grow the specified layers of cells for the wall during the solution process. The cells that are grown will be either prism cells or hex cells, depending on the type of face mesh that is utilized; at no point are the cells visible in the displayed mesh.

Shell conduction can be used to account for thermal mass in transient thermal analysis problems (see Locking the Temperature for Solid and Shell Zones (p. 297) for more information). It can also be used for multiple junctions and allows heat conduction through the junctions. Shell conduction can be applied on boundary walls as well as two-sided walls.

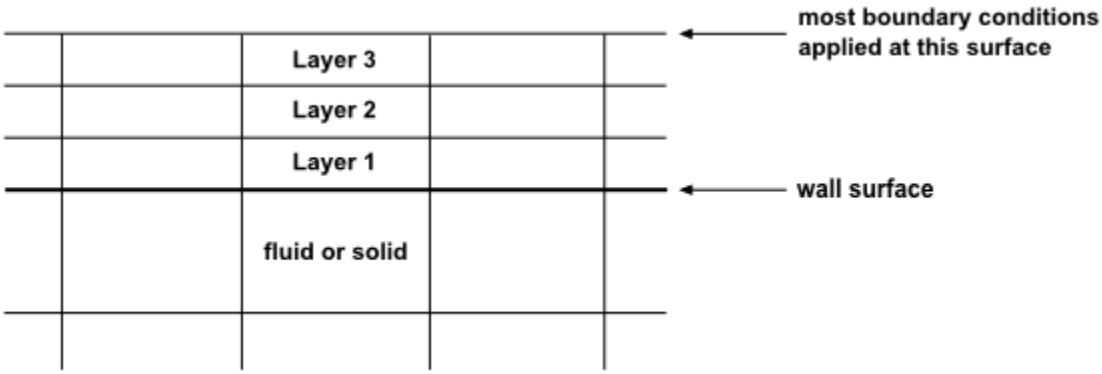
12.2.5.2. Physical Treatment

In the case where shell conduction is applied on a boundary wall, the original surface is referred to as the "wall surface" and is always adjacent to the fluid / solid cells. As part of the solution, layers of wall cells are grown away from the fluid / solid zone. These layers are numbered sequentially, starting with the one closest to the wall surface. See Figure 12.5: A Boundary Wall with Shell Conduction (p. 839) for a visualization of how the solver treats shells.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

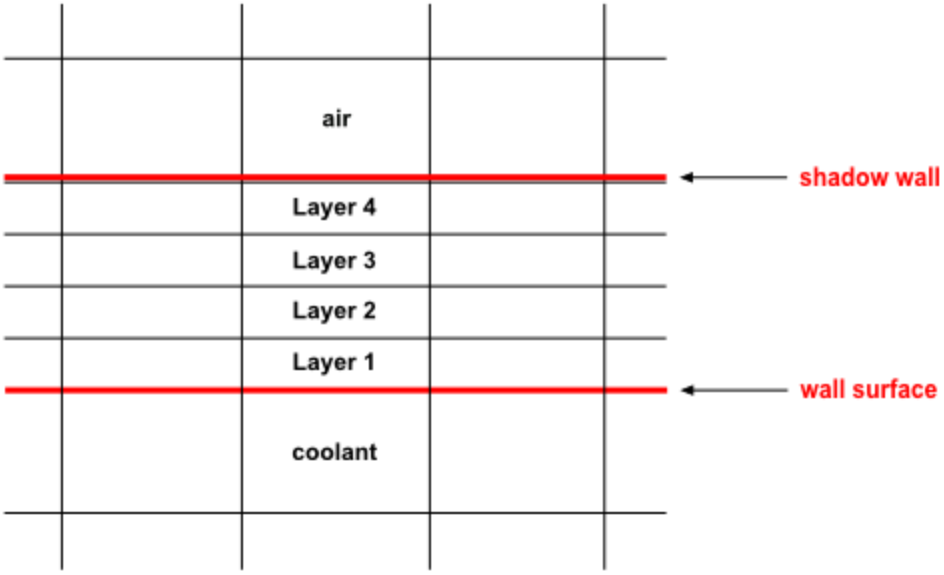
For the most part, the boundary conditions that you specify for the boundary wall are applied to the layer surface that is furthest from the wall surface; only the internal emissivity is applied at the wall surface. The sides of the shell zone require boundary conditions as well. If the shell is connected to another wall that does not have shell conduction enabled, the shell side will take the boundary condition of the attached wall. The sides will be adiabatic if they are connected to face zones that have a boundary condition type other than **wall**. If the attached wall has shell conduction enabled, then the common sides at the junction will be coupled.

Figure 12.5: A Boundary Wall with Shell Conduction



The layers of cells grown for two-sided walls can be visualized in a similar way to Figure 12.5: A Boundary Wall with Shell Conduction (p. 839), except that the layer surface that is furthest from the wall surface is also adjacent to a shadow wall (which is then adjacent to the other fluid / solid zone).

Figure 12.6: A Two-Sided Wall with Shell Conduction



12.2.5.3. Limitations of Shell Conduction Walls

The following is a list of limitations for the shell conduction model:

- Shells cannot be created on non-conformal interfaces, including mapped interfaces.
- Shell conduction cannot be used on moving wall zones.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Shell conduction cannot be used with FMG initialization.
- Shell conduction is not available when the wall is set up to receive thermal data via system coupling.
- Shell conduction is not available for 2D.
- Shell conduction is available only when the pressure-based solver is used.
- Shell conducting walls cannot be split or merged. If you need to split or merge a shell conducting wall, you will need to turn off the **Shell Conduction** option for the wall (in the **Wall** dialog box), perform the split or merge operation, and then enable **Shell Conduction** for the new wall zones.
- The shell conduction model cannot be used on a wall zone that has been adapted. If you want to perform adaption elsewhere in the computational domain, be sure to use the mask register described in Manipulating Adaption Registers (p. 1701). This will ensure that adaption is not performed on the shell conducting wall.
- Fluxes at the ends of a shell conducting wall are not included in heat balance reports. These fluxes are accounted for correctly in the ANSYS Fluent solution, but are not listed in the flux report.
- The junction of a wall with shell conduction enabled and a non-conformal coupled wall is not supported. Such a junction will not be thermally connected, that is, there will be no heat transfer between the shell and the mesh interface wall.
- When running the parallel solver with the shell conduction model, note that coupled walls are encapsulated. If you encounter problems with the partitioning of the mesh, you can try changing the encapsulation method to see if that resolves the problem (see Troubleshooting (p. 2056) for this and other troubleshooting options).
- When running the parallel solver with the shell conduction model, junctions between coupled walls and periodic boundaries are not supported. An error message will be thrown and shell conduction will be disabled.

12.2.5.4. Managing Shell Conduction Walls

The **Shell Conduction Manager** dialog box provides a convenient way for you to manage, define, and display multiple shell conduction zones. You can

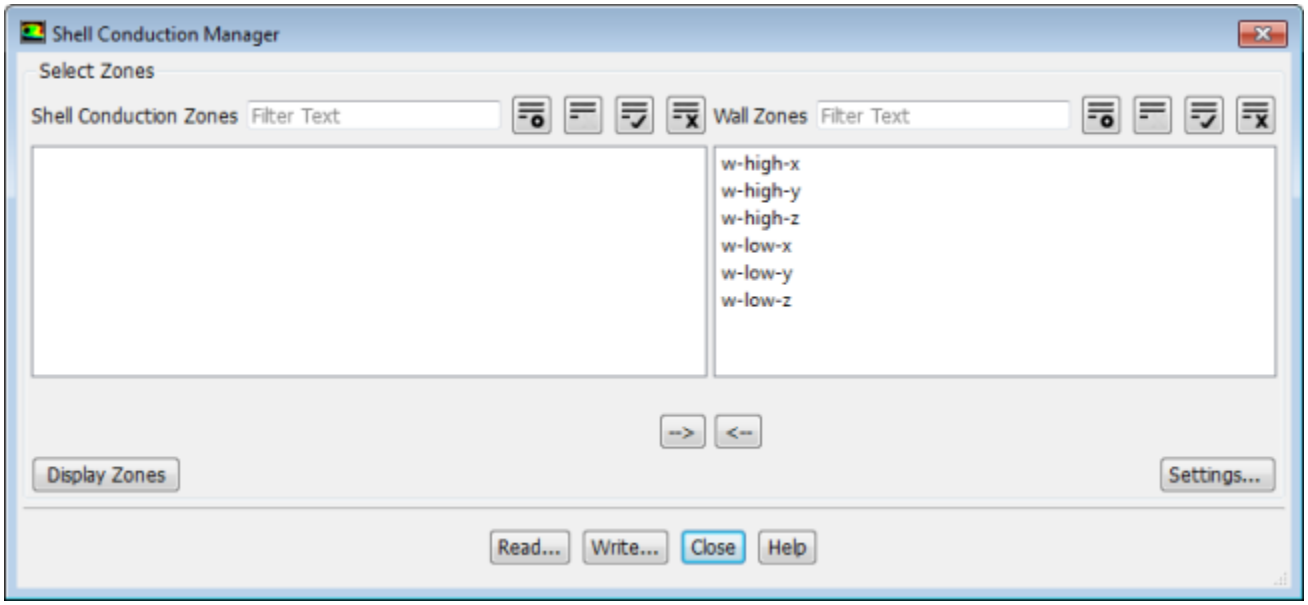
- display any wall zone
- enable or disable shell conduction for a wall
- define the settings for shell conduction walls or layers, such as thickness, material, and heat generation rate

Note that you can still enable shell conduction for a wall and define the related settings using the **Wall** dialog box, as described in Shell Conduction (p. 376). However, the **Shell Conduction Manager** dialog box provides you with an alternative to do such activities from a single dialog box for all the walls, instead of visiting the individual **Wall** boundary condition dialog boxes.

Setting Up Physics → Model Specific → Shell Conduction...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 12.7: The Shell Conduction Manager Dialog Box



To manage shell conduction using the **Shell Conduction Manager** dialog box, perform the following steps:

1. (optional) If you have previously generated a CSV file that contains shell conduction settings for the layers of the walls, you can read it by clicking **Read...** at the bottom of the **Shell Conduction Manager** dialog box and using the dialog box that opens; otherwise, you should proceed to the steps that follow.

   Using a CSV file can be helpful when you have a large number of layers and walls. It is not necessary to create the CSV file from nothing, as you can create a simplified CSV file with sample values using the **Write...** button (as described in step 5.), and then revise the settings using a spreadsheet program. See Shell Conduction Settings File Format (p. 2868) for details about the format of the CSV file.

2. Use the -> and <- buttons to move the selected zones from one list to another, and thereby define whether they are shells. Zones with shell conduction enabled are listed under **Shell Conduction Zones** and those without shell conduction are listed under **Wall Zones**. In essence, the -> button disables shell conduction and the <- button enables shell conduction.

3. Click the **Display Zones** button to display the selected walls in the graphics window. Note that you can select walls with or without shell conduction. The zones will be displayed with different colors depending on the option selected in the **Mesh Colors** dialog box, which is accessible from the **Mesh Display** dialog box.

4. Define settings for all of the shells by performing the following steps:

   a. Select walls from the **Shell Conduction Zones** for which you want to define the same settings.

   b. Click the **Settings...** button to open the **Shell Conduction Layers** dialog box (Figure 12.8: Shell Conduction Layers Dialog Box (p. 842)). There you can specify the number of layers, and then define the thickness, material, and heat generation rate for each layer. Note that you have the option of defining the heat generation rate using a user-defined function (UDF) that utilizes the

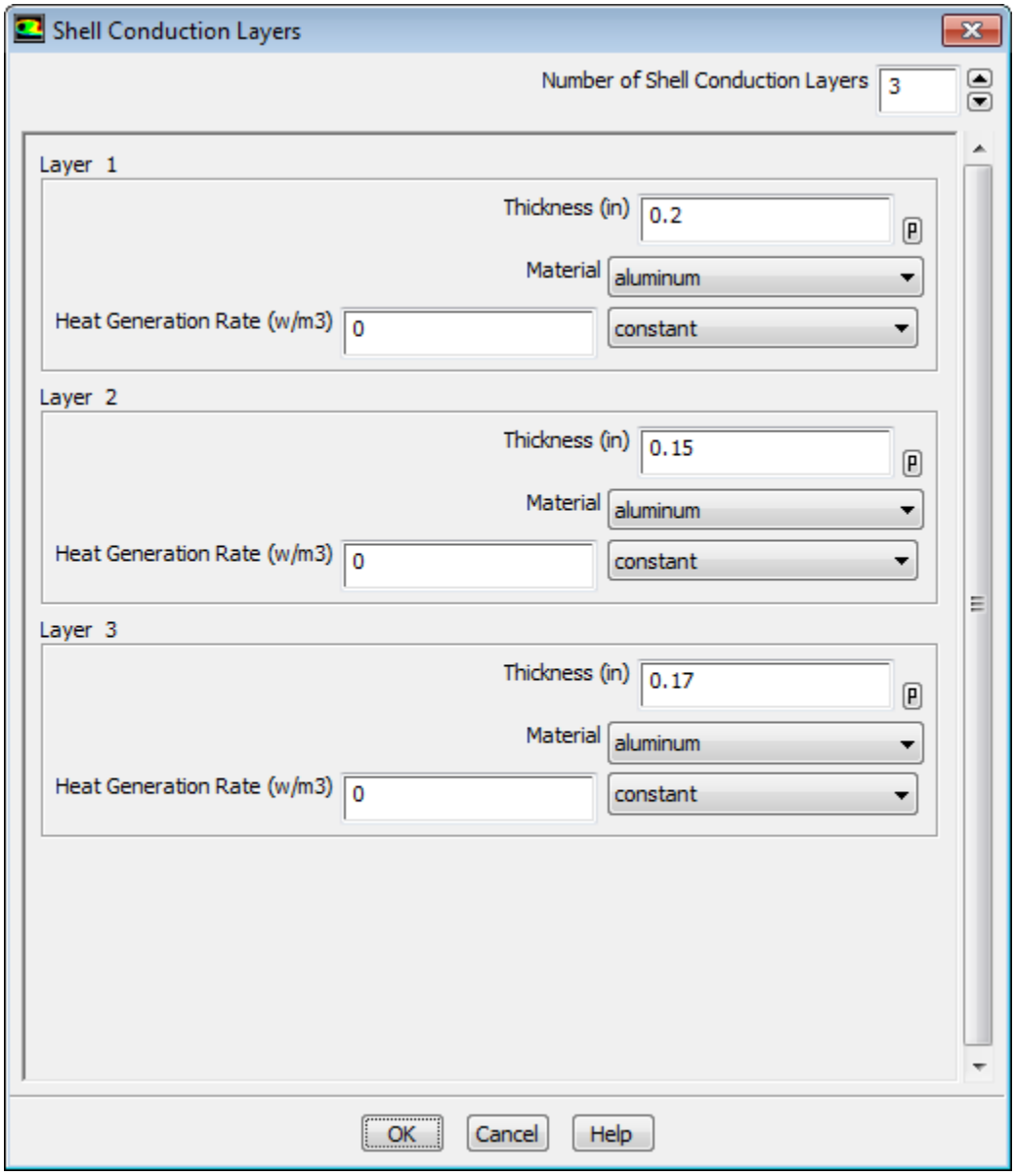
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DEFINE_PROFILE macro; for more information on creating and using user-defined functions, see the Fluent Customization Manual.

Important

You must specify a nonzero-thickness for every layer.

Figure 12.8: Shell Conduction Layers Dialog Box



c. Click **OK** to save your settings for the selected walls and close the **Shell Conduction Layers** dialog box.

5. If you want to save all of the settings you specified using the **Shell Conduction Manager** dialog box as a CSV file, click **Write...** and specify a name in the dialog box that opens. You can edit the shell conduction settings in this CSV file using a spreadsheet program, and then read it in this or a separate case file, as described in step 1. Note that information will not be written for any wall that defines the heat generation rate using a UDF.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that you have the option of disabling shell conduction in every wall with a single action, by using the following text command. This capability is available in both serial and parallel mode.

`define` → `boundary-conditions` → `modify-zones` → `delete-all-shells`

12.2.5.5. Initializing Shells

Shell zones can be patched using the **Patch** dialog box, as described in Patching Values in Selected Cells (p. 1593).

Solution → **Initialization** **Patch...**

12.2.5.6. Locking the Temperature for Shells

You can lock (or "freeze") the temperature values for all the cells in shells and solid zones (including those to which you have a hooked an energy source through a UDF) by using the `solve/set/lock-solid-temperature?` text command, as described in Locking the Temperature for Solid and Shell Zones (p. 297).

12.2.5.7. Postprocessing Shells

In order to facilitate the postprocessing of shells, surfaces (referred to as "shell surfaces") are created at the interface of adjacent layers, as well as at the layer surface that is furthest from the wall surface. Note that shell surfaces are not created on the sides of the shell zones or in the interior of the layers. The shell surfaces can then be selected from the **Surfaces** list available in a variety of dialog boxes (for example, the **Contours** dialog box, the **Surface Report Definition** dialog box, and the **Solution XY Plot** dialog box) in the same way as the surfaces for the wall and (for two-sided walls) the shadow wall. When visualized in the graphics display window, these shell surfaces will be coincident with the wall they are associated with.

The default naming convention for these shell surfaces is <*wall_name*>**-**<*c0*>**:**<*c1*>, where:

- <*wall_name*> is the name of the wall on which the layers have been grown.
- <*c0*> is the rank of the layer on the c0 side of the interface (that is, the layer closest to the wall surface).
- <*c1*> is the rank of the layer on the c1 side of the interface (that is, the layer furthest from the wall surface). Note that the layer surface that is furthest from the wall surface does not have a layer on the c1 side, and so a different convention is used: for boundary walls, <*c1*> is set to **external**; for two-sided walls, <*c1*> is set to the name of the fluid / solid adjacent to the shadow wall.

You have the ability to rename the shell surfaces, if you want. The following figures illustrate the default naming convention, where the wall/shadow surfaces are displayed in red and the shell surfaces are displayed in black.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 12.9: Shell Surface Names for a Boundary Wall

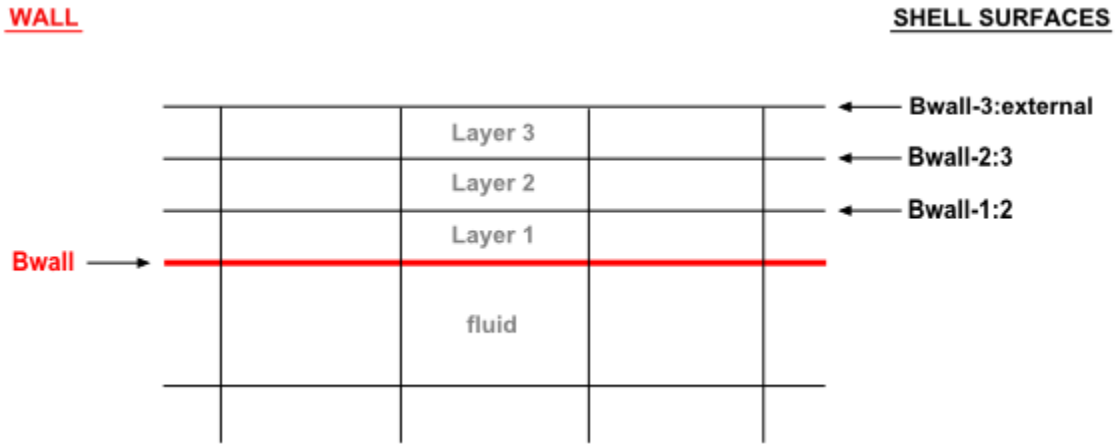
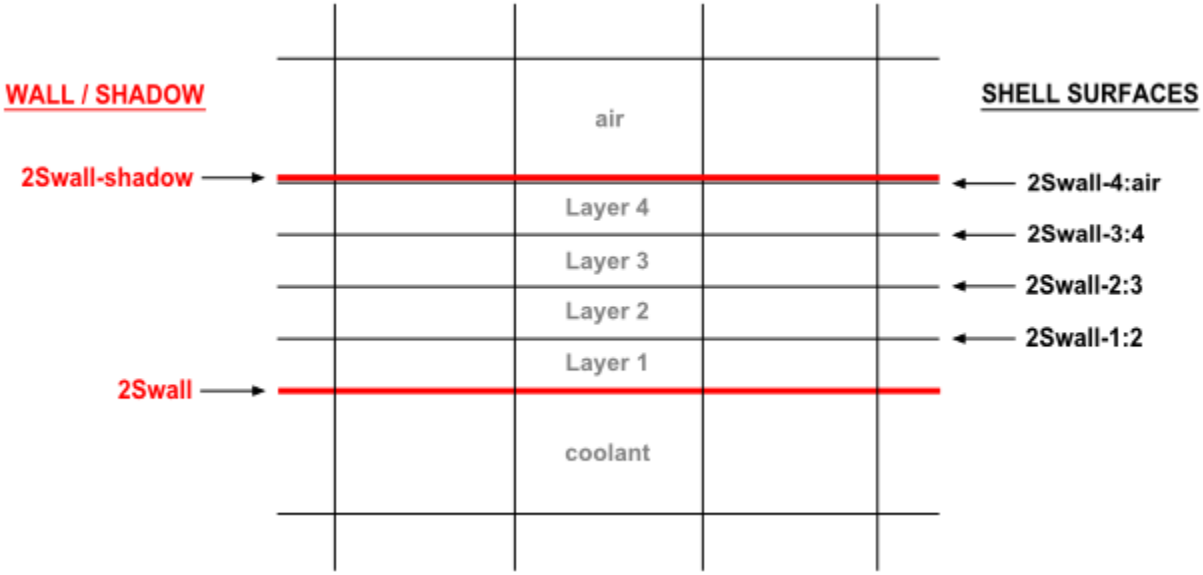


Figure 12.10: Shell Surface Names for a Two-Sided Wall



You can use the walls, shadow walls, and shell surfaces to display the temperature of the interfaces and adjacent cells. The **Temperature...** category provides two options: the temperature of the cell on the c0 side of the surface is stored as **Static Temperature**; and the temperature of the surface itself is stored as **Wall Temperature**. If a more detailed analysis of the solid zone and surfaces is required, then you should consider creating layers of solid cells in your meshing application.

The following examples illustrate how to postprocess the surfaces shown in Figure 12.9: Shell Surface Names for a Boundary Wall (p. 844):

- The **Static Temperature** and **Wall Temperature** of **Bwall** will provide the temperature of the fluid cells adjacent to **Bwall** and the temperature of the **Bwall** surface itself, respectively.
- The **Static Temperature** and **Wall Temperature** of **Bwall-1:2** will provide the temperature of the cells of **Layer 1** and the temperature of the interface between **Layer 1** and **Layer 2** (that is, the **Bwall-1:2** shell surface), respectively.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The **Static Temperature** and **Wall Temperature** of **Bwall-3:external** will provide the temperature of the cells of **Layer 3** and the temperature of the layer surface that is furthest from the wall surface (that is, the **Bwall-3:external** shell surface), respectively.

The following examples illustrate how to postprocess the surfaces shown in Figure 12.10: Shell Surface Names for a Two-Sided Wall (p. 844):

- The **Static Temperature** and **Wall Temperature** of **2Swall-4:air** will provide the temperature of the cells of **Layer 4** and the temperature of the interface between **Layer 4** and the shadow wall (that is, the **2Swall-4:air** shell surface), respectively.

- The **Static Temperature** and **Wall Temperature** of **2Swall-shadow** will provide the temperature of the air cells adjacent to **2Swall-shadow** and the temperature of the shadow wall, respectively.

When postprocessing shells, note the following limitations:

- You should not enable **Node Values** when postprocessing any surfaces associated with shells (that is, the wall, shadow wall, or shell surfaces). The layers share nodes at the edges, and thus the averaging at such nodes is erroneous.

- You cannot postprocess shells in ANSYS CFD-Post, as shell surfaces are not supported.

12.3. Modeling Radiation

Information about radiation modeling is presented in the following sections:

12.3.1. Using the Radiation Models
12.3.2. Setting Up the P-1 Model with Non-Gray Radiation
12.3.3. Setting Up the DTRM
12.3.4. Setting Up the S2S Model
12.3.5. Setting Up the DO Model
12.3.6. Setting Up the MC Model
12.3.7. Defining Material Properties for Radiation
12.3.8. Defining Boundary Conditions for Radiation
12.3.9. Solution Strategies for Radiation Modeling
12.3.10. Postprocessing Radiation Quantities
12.3.11. Solar Load Model

For theoretical information about the radiation models in ANSYS Fluent, refer to Modeling Radiation in the Theory Guide.

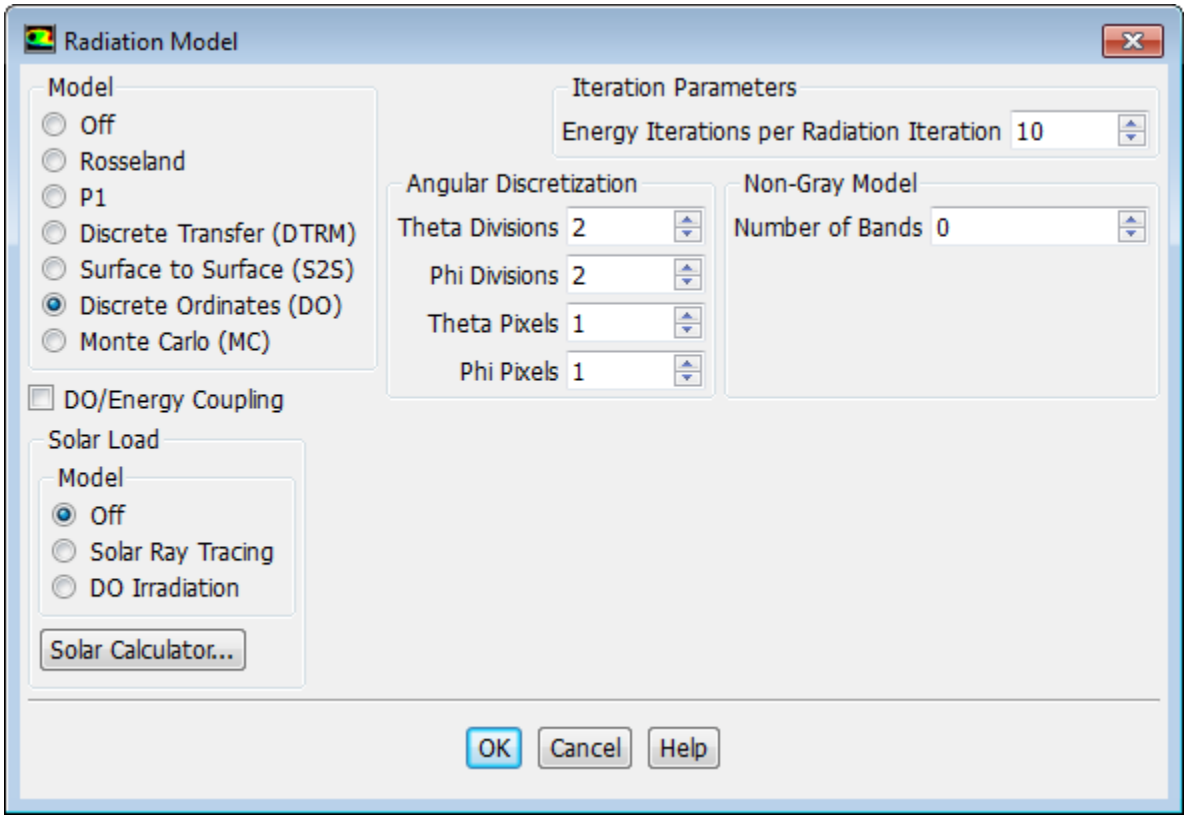
12.3.1. Using the Radiation Models

The procedure for setting up and solving a radiation problem is outlined below, and described in detail in referenced sections. Steps that are relevant only for a particular radiation model are noted as such. The steps that are pertinent to radiation modeling, are shown here. For information about inputs related to other models that you are using in conjunction with radiation, see the appropriate sections for those models.

1. Enable radiative heat transfer by selecting a radiation model (**Rosseland**, **P1**, **Discrete Transfer (DTRM)**, **Surface to Surface (S2S)**, **Discrete Ordinates**, or **Monte Carlo (MC)**) under **Model** in the **Radiation Model** dialog box (Figure 12.11: The Radiation Model Dialog Box (DO Model) (p. 846)).

   Setup → **Models** → **Radiation** → **Edit...**

Figure 12.11: The Radiation Model Dialog Box (DO Model)



Important

The **Rosseland** model can be used only with the pressure-based solver.

Note that when the P-1, the DTRM, the S2S, DO, or MC model is enabled, the **Radiation Model** dialog box expands to show additional parameters. These parameters will not appear if you select the Rosseland model. If you are running a 3D case, you will have the added option of using the solar load model. The solar load options will be displayed in the dialog box, below the radiation model settings.

When the radiation model is active, the radiation fluxes will be included in the solution of the energy equation at each iteration. If you set up a problem with the radiation model turned on, and you then decide to turn it off completely, you must select the **Off** button in the **Radiation Model** dialog box.

Note that when you enable a radiation model, ANSYS Fluent will automatically enable the energy equation so that step is not needed.

2. Set the appropriate radiation parameters.

   a. If you are modeling non-gray radiation using the P-1 model, define the non-gray radiation parameters as described in Setting Up the P-1 Model with Non-Gray Radiation (p. 847).

   b. If you are using the DTRM, define the ray tracing as described in Setting Up the DTRM (p. 848).

   c. If you are using the S2S model, define the surface clusters and view factors settings and compute or read the view factors as described in Setting Up the S2S Model (p. 851).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

d. If you are using the DO model, choose **DO/Energy Coupling** if desired, define the angular discretization as described in Setting Up the DO Model (p. 864) and, if relevant, define the non-gray radiation parameters as described in Defining Non-Gray Radiation for the DO Model (p. 865).

e. If you are using the MC model, specify the **Target Number of Histories** as described in Setting Up the MC Model (p. 867) and, if relevant, similar to when using the DO model, define the non-gray radiation parameters as described in Defining Non-Gray Radiation for the DO Model (p. 865).

3. Define the material properties as described in Defining Material Properties for Radiation (p. 868).

4. Define the boundary conditions as described in Defining Boundary Conditions for Radiation (p. 869). If your model contains a semi-transparent medium, see the information below on setting up semi-transparent media.

5. Set the parameters that control the solution (DTRM, DO, MC, S2S, and P-1 only) as described in Solution Strategies for Radiation Modeling (p. 878).

6. Run the solution as described in Running the Calculation (p. 880).

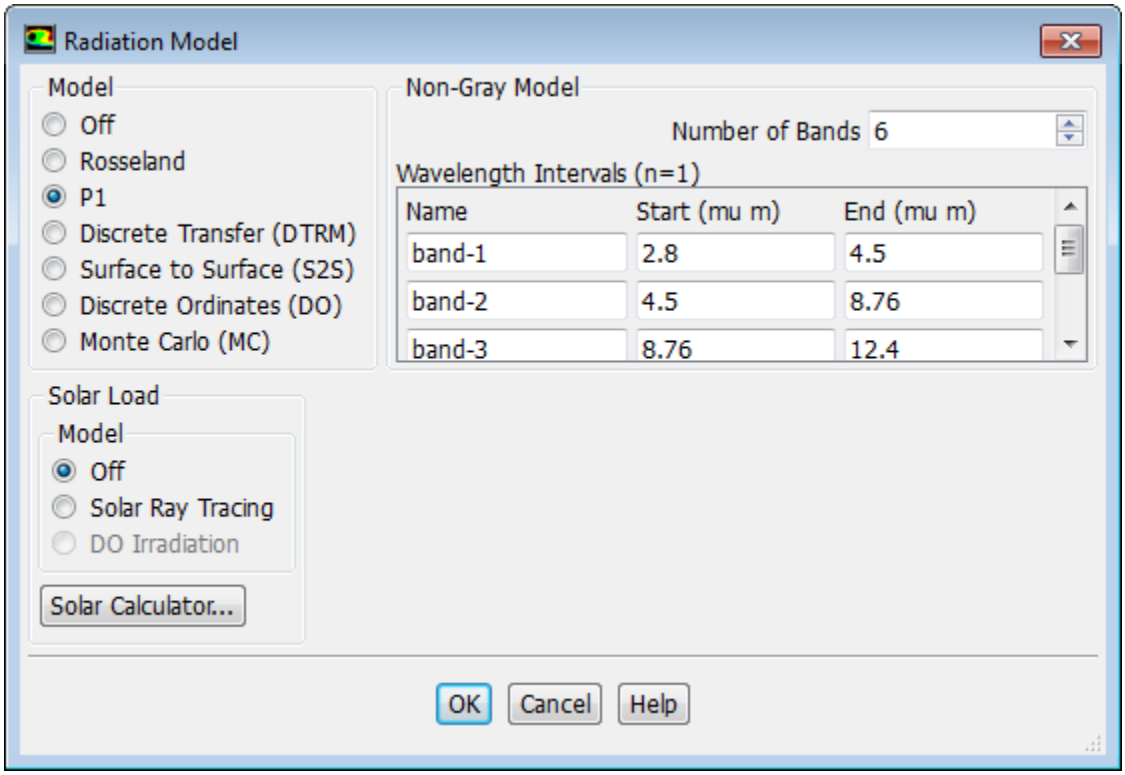
7. Postprocess the results as described in Postprocessing Radiation Quantities (p. 882).

12.3.2. Setting Up the P-1 Model with Non-Gray Radiation

If you want to model non-gray radiation using the P-1 model, you can specify the **Number of Bands** under **Non-Gray Model** in the expanded **Radiation Model** dialog box (Figure 12.12: The Radiation Model Dialog Box (Non-Gray P-1 Model) (p. 848)). By default, the **Number of Bands** is set to zero, indicating that only gray radiation will be modeled. Because the cost of computation increases directly with the number of bands, you should try to minimize the number of bands used. When a nonzero **Number of Bands** is specified, the **Radiation Model** dialog box will expand once again to show the **Wavelength Intervals**. For each wavelength band, you can specify a **Name**, as well as the **Start** and **End** wavelength of the band in μm. Note that the wavelength bands are specified for vacuum (n=1). For more information about non-gray radiation calculations, see Defining Non-Gray Radiation for the DO Model (p. 865).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 12.12: The Radiation Model Dialog Box (Non-Gray P-1 Model)



ANSYS Fluent allows you to use a user-defined function (UDF) to modify the emissivity weighting factor $F\left(0\rightarrow n\lambda_2 T\right)-F\left(0\rightarrow n\lambda_1 T\right)$ (which otherwise defaults to the black body emission factor obtained from a standard Planck distribution). The emissivity weighting factor appears in the emission term of the radiative transfer equation for the non-gray model, as shown in Equation 5.25 in the Theory Guide. For more information, see the Fluent Customization Manual.

12.3.3. Setting Up the DTRM

For information about setting up the DTRM, see the following sections:

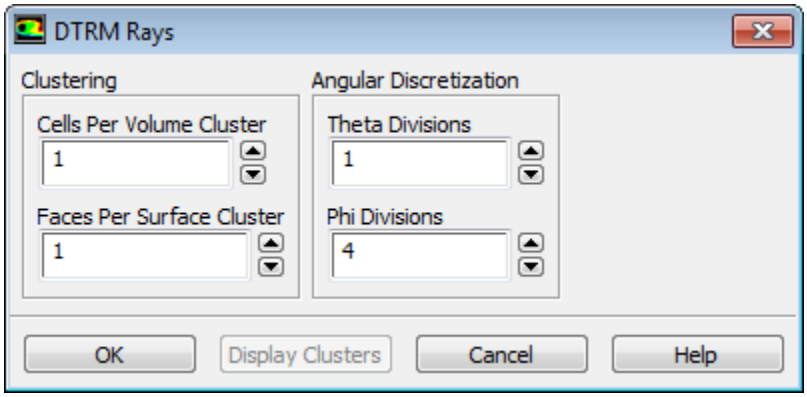
12.3.3.1. Defining the Rays
12.3.3.2. Controlling the Clusters
12.3.3.3. Controlling the Rays
12.3.3.4. Writing and Reading the DTRM Ray File
12.3.3.5. Displaying the Clusters

12.3.3.1. Defining the Rays

When you select the **Discrete Transfer** model and click **OK** in the **Radiation Model** dialog box, the **DTRM Rays** dialog box (Figure 12.13: The DTRM Rays Dialog Box (p. 849)) will open automatically. If you need to modify the current settings later in the problem setup or solution procedure, you can open this dialog box by clicking **DTRM Rays...** in the **Setting Up Physics** ribbon tab (**Model Specific** group box).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 12.13: The DTRM Rays Dialog Box



In this dialog box you will set parameters for and create the rays and clusters discussed in The DTRM Equations in the Theory Guide.

The procedure is as follows:

1. To control the number of radiating surfaces and absorbing cells, set the **Cells Per Volume Cluster** and **Faces Per Surface Cluster**. (See the explanation below.)

2. To control the number of rays being traced, set the number of **Theta Divisions** and **Phi Divisions**. (Guidelines are provided below.)

3. When you click **OK** in the **DTRM Rays** dialog box, The Select File Dialog Box (p. 19) will open prompting you for the name of the "ray file". After you have specified the filename and chosen whether to write a binary ray file, ANSYS Fluent will write the ray file and then read it afterward. During the write process the status of the DTRM ray tracing will be reported in the ANSYS Fluent console. For example:

   ```
   Completed 25% tracing of DTRM rays

   Completed 50% tracing of DTRM rays

   Completed 75% tracing of DTRM rays

   Completed 100 % tracing of DTRM rays
   ```

   See following sections for details on **DTRM Rays** dialog box inputs.

Important

If you cancel the **DTRM Rays** dialog box without writing and reading the ray file, the DTRM will be disabled.

12.3.3.2. Controlling the Clusters

Your inputs for **Cells Per Volume Cluster** and **Faces Per Surface Cluster** will control the number of radiating surfaces and absorbing cells. By default, each is set to 1, so the number of surface clusters (radiating surfaces) will be the number of boundary faces, and the number of volume clusters (absorbing cells) will be the number of cells in the domain. For small 2D problems, these are acceptable numbers,

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

but for larger problems you will want to reduce the number of surface and/or volume clusters in order to reduce the ray-tracing expense. See Clustering in the Theory Guide for details about clustering.

Note

- Only zones attached to fluid cell zones are used for creating surface clusters.
- **Faces Per Surface Cluster** (FPSC) is indicative of the actual reduction factor except when the surfaces are planar. The surface clustering algorithm uses other criteria (angle between faces, and so on) to ensure the clusters are nearly planar, so it is possible that there will be no further reduction in the total number of clusters beyond a certain value of FPSC.

12.3.3.3. Controlling the Rays

Your inputs for **Theta Divisions** and **Phi Divisions** will control the number of rays being traced from each surface cluster (radiating surface).

Theta Divisions defines the number of discrete divisions in the angle θ used to define the solid angle about a point P on a surface. The solid angle is defined as θ varies from 0 to 90 degrees (in the Theory Guide), and the default setting of 1 for the number of discrete settings implies that there will be one ray traced from the surface.

Phi Divisions defines the number of discrete divisions in the angle ϕ used to define the solid angle about a point P on a surface. The solid angle is defined as ϕ varies from 0 to 360 degrees (Figure 5.2: Angles θ and ϕ Defining the Hemispherical Solid Angle About a Point P in the Theory Guide). The default setting of 4 implies that each ray traced from the surface will be located at a 90° angle, and in combination with the default setting for **Theta Divisions**, above, implies that 4 rays will be traced from each surface control volume. In many cases, it is recommended that you at least double the number of divisions in θ and ϕ.

12.3.3.4. Writing and Reading the DTRM Ray File

After you have activated the DTRM and defined all of the parameters controlling the ray tracing, you must create a ray file, which will be read back in and used during the radiation calculation. The ray file contains a description of the ray traces (for example, path lengths, cells traversed by each ray). This information is stored in the ray file, instead of being recomputed, in order to speed up the calculation process.

By default, a binary ray file will be written. You can also create text (formatted) ray files by turning off the **Write Binary Files** option in The Select File Dialog Box (p. 19).

Important

Do not write or read a compressed ray file, because ANSYS Fluent will not be able to access the ray tracing information properly from a compressed ray file.

The ray filename must be specified to ANSYS Fluent only once. Thereafter, the filename is stored in your case file and the ray file will be automatically read into ANSYS Fluent whenever the case file is read. ANSYS Fluent will remind you that it is reading the ray file after it finishes reading the rest of the case file by reporting its progress in the console.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that the ray filename stored in your case file may not contain the full name of the directory in which the ray file exists. The full directory name will be stored in the case file only if you initially read the ray file through the GUI (or if you typed in the directory name along with the filename when using the text interface). In the event that the full directory name is absent, the automatic reading of the ray file may fail (since ANSYS Fluent does not know in which directory to look for the file), and you will need to manually specify the ray file, using the **File/Read/DTRM Rays...** ribbon tab item. The safest approaches are to use the GUI when you first read the ray file or to supply the full directory name when using the text interface.

Important

You must recreate the ray file whenever you do anything that changes the mesh, such as:

- change the type of a boundary zone
- adapt the mesh
- scale the mesh

You can open the **DTRM Rays** dialog box directly by clicking **DTRM Rays...** in the **Setting Up Physics** ribbon tab (**Model Specific** group box).

12.3.3.5. Displaying the Clusters

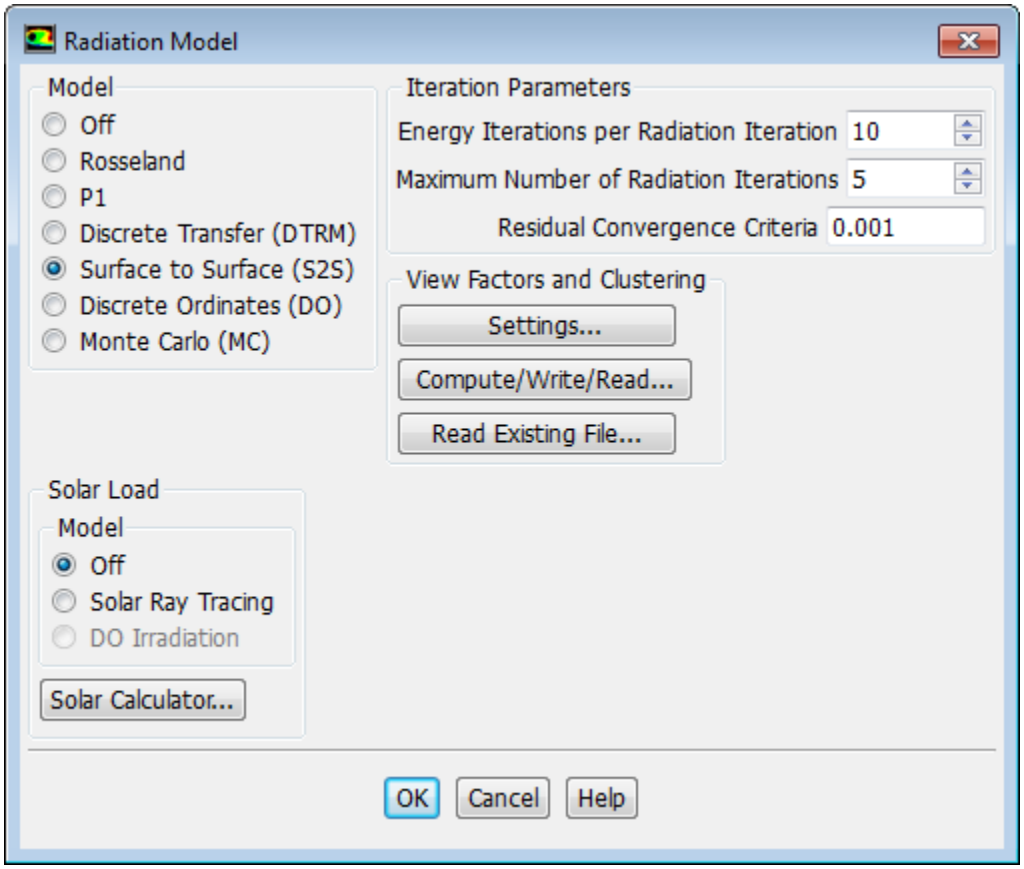
Once a ray file has been created or read in manually, you can click the **Display Clusters** button in the **DTRM Rays** dialog box to graphically display the clusters in the domain. See Displaying Rays and Clusters for the DTRM (p. 883) for additional information about displaying rays and clusters.

12.3.4. Setting Up the S2S Model

When you select the **Surface to Surface (S2S)** model, the **Radiation Model** dialog box will expand to show additional parameters (see Figure 12.14: The Radiation Model Dialog Box (S2S Model) (p. 852)). In these additional group boxes, you will set the solution parameters (see S2S Solution Parameters (p. 880) for further details), access the view factors and clustering settings, and compute the view factors for your problem or read previously computed view factors into ANSYS Fluent.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 12.14: The Radiation Model Dialog Box (S2S Model)



The S2S radiation model is very expensive in terms of computation effort and memory requirements when there are a large number of radiating surfaces. You can reduce the number of radiating surfaces by grouping faces together to form surface clusters. The surface cluster information (coordinates and connectivity of the nodes, surface cluster IDs) is used by ANSYS Fluent in the radiosity calculations and to compute the view factors.

Important

- You must recreate the surface cluster information whenever you do anything that changes the mesh, such as:
  - change the type of a boundary zone
  - scale the mesh

  Note that you do *not* need to recalculate view factors after shell conduction at any wall has been enabled or disabled (see Thermal Boundary Conditions at Walls (p. 370) for more information about shell conduction). However, enabling or disabling shell conduction in the parallel version of Fluent does cause a migration of cells and hence a change in the partition. Whenever there is cell migration, the previously read view factor file is no longer valid, and the view factor file must be read again using either the **Read Existing File** button in the **Radiation Model** dialog box or the **File/Read/View Factors...** ribbon tab item.

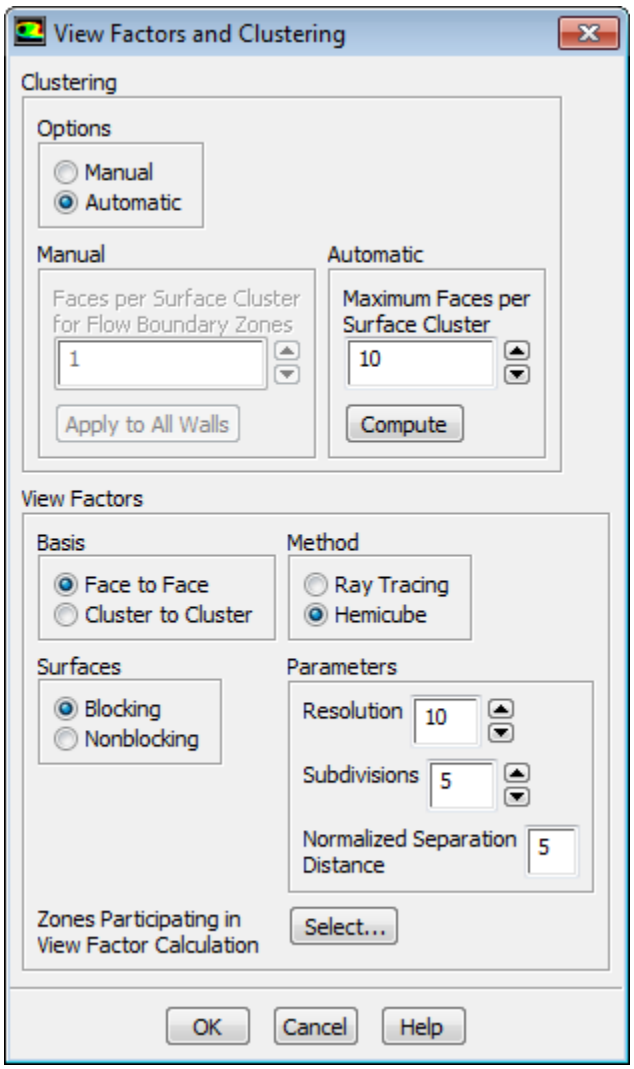
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- ANSYS Fluent will warn you to recreate the cluster/view factor file if a boundary zone has been changed from a wall to an internal wall (or vice versa), or if a boundary zone has been merged, separated, or fused.

12.3.4.1. View Factors and Clustering Settings

You can use the **View Factors and Clustering** dialog box (Figure 12.15: The View Factors and Clustering Dialog Box (p. 853)) to define how the surface clusters are formed and how the view factors are calculated for the S2S model. To open this dialog box, click **Settings...** in the **View Factors and Clustering** group box in the Radiation Model Dialog Box (p. 2190) (Figure 12.14: The Radiation Model Dialog Box (S2S Model) (p. 852)) or use the **File/Write/Surface Clusters...** ribbon tab item.

Figure 12.15: The View Factors and Clustering Dialog Box



12.3.4.1.1. Forming Surface Clusters

You can set the number of faces per surface cluster (FPSC) for each flow boundary (that is, exhaust fan, inlet vent, intake fan, outlet vent, mass-flow inlet, mass-flow outlet, pressure far-field, pressure inlet, pressure outlet, outflow, and velocity inlet boundary) and wall that is adjacent to a fluid zone; in this way, you can control the number of radiating surfaces and (if you select **Cluster to Cluster** for **Basis**)

view factor surfaces. By default, the FPSC value is set to 1 everywhere, so the number of surface clusters (radiating/view factor surfaces) will be equal to the number of boundary faces. For small 2D problems, this is an acceptable number. For larger problems, you may want to reduce the number of surface clusters (that is, increase the FPSC) to reduce both the size of the view factor file and the memory requirement. Such a reduction in the number of clusters, however, comes at the cost of some accuracy. Note that for a thermally coupled mesh interface (that is, a fluid-fluid interface with **Coupled Wall** enabled, or an interface involving a solid zone), the FPSC is always set to 1. See Clustering in the Theory Guide for details about clustering.

Important

If you plan to use the cluster to cluster basis for the view factor calculation, be sure that the FPSC values are appropriate for both the radiosity calculation and the view factors.

Note that during the creation of a cluster on a non-conformal interface, its parent thread is taken into the consideration; it may happen that the cluster is somewhat larger than its child face, which may lead some inaccuracy in the flux calculation.

The **Clustering** group box in the **View Factors and Clustering** dialog box (Figure 12.15: The View Factors and Clustering Dialog Box (p. 853)) provides two methods for revising the default FPSC values:

- manual
- automatic

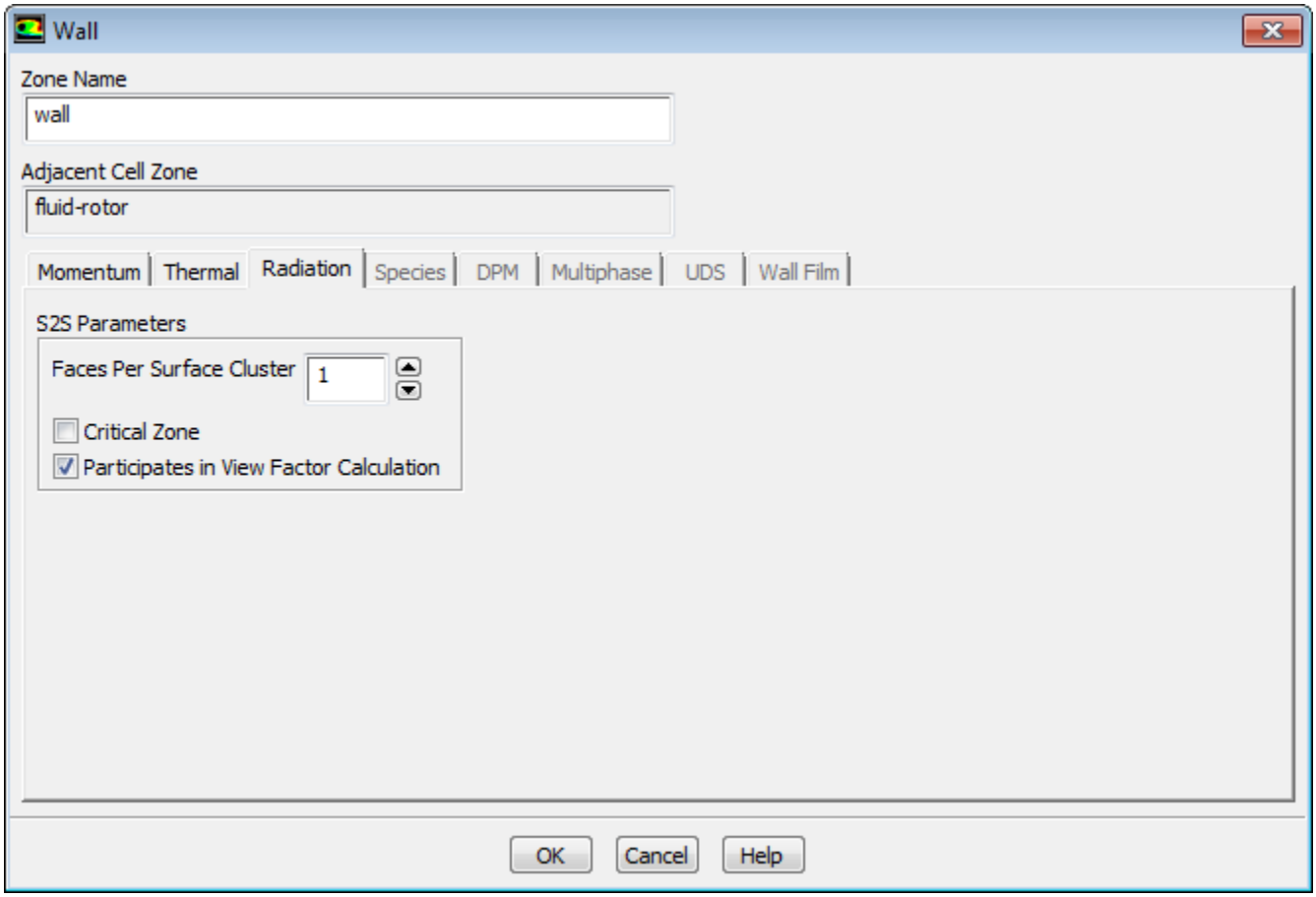
If you select **Manual** in the **Options** group box, you can specify an FPSC value for all flow boundaries in the **Faces per Surface Cluster for Flow Boundary Zones** number-entry box in the **Manual** group box. If you then click **Apply to All Walls**, you will apply this value to all wall zones that are adjacent to fluid zones as well. For a wall that requires a different FPSC value (that is, you may want a lower value for walls in critical areas, and higher values in non-critical areas), you will need to open the boundary condition dialog box (Figure 12.16: The Wall Dialog Box (p. 855)) for that particular wall and modify the **Faces Per Surface Cluster** in the **S2S Parameters** group box of the **Radiation** tab. Note that the **Radiation** tab also allows you to specify whether the wall participates in the view factor calculation, as described in Specifying Boundary Zone Participation (p. 858).

Note

- Only zones attached to fluid cell zones are used for creating surface clusters.
- **Faces Per Surface Cluster** (FPSC) is indicative of the actual reduction factor except when the surfaces are planar. The surface clustering algorithm uses other criteria (angle between faces, and so on) to ensure the clusters are nearly planar, so it is possible that there will be no further reduction in the total number of clusters beyond a certain value of FPSC.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 12.16: The Wall Dialog Box



Important

The **Faces Per Surface Cluster** number-entry box will not be visible in the GUI on wall boundary zones that are attached to a solid.

Using the manual method to specify the FPSC values for walls can become very cumbersome if the model involves a large number of radiating faces, which is typically the case in underhood models. In such circumstances, it is recommended that you use the automatic clustering method instead. In this method, different FPSC values are assigned to the walls automatically, based on the distance of the walls from and the FPSC values of the walls that are defined as critical. The steps you will need to take are as follows:

1. Select **Automatic** from the **Options** list in the **View Factors and Clustering** dialog box.
2. For each wall that you deem critical, perform the following actions in the **Wall** dialog box (Figure 12.16: The Wall Dialog Box (p. 855)):
   a. Click the **Radiation** tab.
   b. Enter an appropriate value for **Faces Per Surface Cluster** in the **S2S Parameters** group box.
   c. Enable the **Critical Zone** option.
   d. Click **OK** to close the **Wall** dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Enter the **Maximum Faces per Surface Cluster** value in the **View Factors and Clustering** dialog box and click the **Compute** button. ANSYS Fluent will automatically calculate and update the **Faces Per Surface Cluster** values for all **Wall** dialog boxes adjacent to fluid zones that do not have **Critical Zone** enabled, without computing the clusters.

4. You can check the automatically assigned FPSC values by opening the boundary condition dialog box of any non-critical wall of interest and examining the value for **Faces Per Surface Cluster** in the **S2S Parameters** group box of **Radiation** tab. You can manually modify the value for **Faces Per Surface Cluster** as necessary.

12.3.4.1.1.1. Setting the Split Angle for Clusters

Whether you set the FPSC value manually or automatically, you have the option of modifying the cutoff or "split" angle between adjacent face normals for the purpose of controlling surface clustering. The split angle sets the limit for which adjacent faces are clustered. A smaller split angle allows for a better representation of the view factor. By default, no surface cluster will contain any face that has a face normal greater than 20°. To modify the value of this parameter, you can use the `split-angle` text command:

`define` → `models` → `radiation` → `s2s-parameters` → `split-angle`

or

`file` → `write-surface-clusters` → `split-angle`

12.3.4.1.2. Setting Up the View Factor Calculation

You can control many aspects of how the view factors are calculated for your S2S model: how surfaces are defined; the computational method and related parameters; whether surface blocking will be accounted for; and which boundary zones will participate in the calculation. All of these controls are available in the **View Factors** group box in the **View Factors and Clustering** dialog box (Figure 12.15: The View Factors and Clustering Dialog Box (p. 853)), and are described in the sections that follow.

12.3.4.1.2.1. Selecting the Basis for Computing View Factors

ANSYS Fluent allows two ways to define the surfaces used for the view factor calculation, as described in Clustering and View Factors in the Theory Guide. If you want the surfaces to be the boundary faces of the mesh, select **Face to Face** for **Basis** in the **View Factor** group box of the **View Factors and Clustering** dialog box. This is the default selection.

Alternatively, you can select **Cluster to Cluster** for **Basis**, in order to reduce the computational expense and storage requirements. In this case, the surfaces used to calculate the view factors are the clusters defined by the settings in the **Cluster** group box of the **View Factors and Clustering** dialog box. The reduction in computational time will be proportional to the number of surface clusters used in the view factor calculation. The trade-offs of using the cluster to cluster basis for the calculation are that the accuracy may decrease, and the following limitations apply:

- The mesh must be 3D.
- You cannot subdivide the faces as part of the hemicube method parameters, which can cause the view factors to be overestimated.
- Polyhedral meshes are restricted to 1 face per surface cluster.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When using the cluster to cluster basis with non-conformal interfaces, the **Faces Per Surface Cluster** will be limited to one, giving no advantage to the cluster to cluster basis.

12.3.4.1.2.2. Selecting the Method for Computing View Factors

ANSYS Fluent provides two methods for computing view factors: the ray tracing method (which is selected by default) and the hemicube method. The following limitations apply:

- The hemicube method is available only for 3D and axisymmetric cases.
- The hemicube method should not be used when any of the zones are defined as periodic or symmetry boundaries, as these types are not currently supported.
- For the ray tracing method, when running in distributed parallel mode, the working directory must be shared across all machines.

The hemicube method uses a differential area-to-area method and calculates the view factors on a row-by-row basis. The view factors calculated from the differential areas are summed to provide the view factor for the whole surface. This method originated from the use of the radiosity approach in the field of computer graphics [22] (p. 2888).

To use the hemicube method to compute the view factors, select **Hemicube** from the **Method** list in the **View Factors and Clustering** dialog box. It is recommended that you use the hemicube method for large, complex models with few obstructing surfaces between the radiating surfaces.

The hemicube method is based upon three assumptions about the geometry of the surfaces: aliasing, visibility, and proximity. To validate these assumptions, you can specify three different hemicube parameters, which can help you obtain better accuracy in calculating view factors. In most cases, however, the default settings will be sufficient.

- Aliasing—The true projection of each visible face onto the hemicube can be accurately accounted for by using a finite-resolution hemicube. As described previously, the faces are projected onto a hemicube. Because of the finite resolution of the hemicube, the projected areas and resulting view factors may be overestimated or underestimated. Aliasing effects can be reduced by increasing the value of the **Resolution** of the hemicube in the **Parameters** group box.
- Visibility—The visibility between any two faces does not change. In some cases, face i has a complete view of face k from its centroid, but some other face j occludes much of face k from face i. In such a case, the hemicube method will overestimate the view factor between face i and face k calculated from the centroid of face i. This error can be reduced by subdividing face i into smaller subfaces. You can specify the number of subfaces by entering a value for **Subdivisions** in the **Parameters** group box. Note that you cannot subdivide the faces when **Cluster to Cluster** is selected for **Basis**.
- Proximity—The distance between faces is great compared to the effective diameter of the faces. The proximity assumption is violated whenever faces are close together in comparison to their effective diameter or are adjacent to one another. In such cases, the distances between the centroid of one face and all points on the other face vary greatly. Since the view factor dependence on distance is nonlinear, the result is a poor estimate of the view factor.

  In the **Parameters** group box, you can set a limit for the **Normalized Separation Distance**, which is the ratio of the minimum face separation to the effective diameter of the face. If the computed normalized separation distance is less than the specified value, the face will then be divided into a number of subfaces until the normalized distances of the subfaces are greater than the specified

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

value. Alternatively, you can specify the number of subfaces to create for such faces by entering a value for **Subdivisions**.

While the hemicube method projects radiating surfaces onto a hemicube, the ray tracing method instead traces rays through the centers of every hemicube face to determine which surfaces are visible through that face. Note that the ray tracing method does not subdivide the faces (as can be done when using the hemicube method by setting the **Subdivisions** or **Normalized Separation Distance** parameters), and so the view factors may be less accurate than those calculated using the hemicube method for surfaces that have a normalized separation distance less than 5.

To use the ray tracing method to compute the view factors, select **Ray Tracing** from the **Method** list in the **View Factors and Clustering** dialog box. You can adjust the value of the **Resolution** in the **Parameters** group box in order to reduce the impact of aliasing effects, as described previously.

12.3.4.1.2.3. Accounting for Blocking Surfaces

View factor calculations depend on the geometric orientations of surface pairs with respect to each other. Two situations may be encountered when examining surface pairs:

- If there is no obstruction between the surface pairs under consideration, then they are referred to as "non-blocking" surfaces.
- If there is another surface blocking the views between the surfaces under consideration, then they are referred to as "blocking" surfaces. Blocking will change the view factors between the surface pairs and require additional checks to compute the correct value of the view factors.

For cases with blocking surfaces, select **Blocking** from the **Surfaces** list in the **View Factors and Clustering** dialog box. For cases with nonblocking surfaces, you can choose either **Blocking** or **Nonblocking** without affecting the accuracy. However, it is better to choose **Nonblocking** for such cases, as it takes less time to compute.

12.3.4.1.2.4. Specifying Boundary Zone Participation

You can choose to exclude walls and inlet and exit boundaries from participating in the view factor calculation. If you are unsure whether it is necessary to calculate view factors for a particular boundary zone ahead of time, it is recommended that you allow it to participate; you can always reverse this decision as part of a future calculation run.

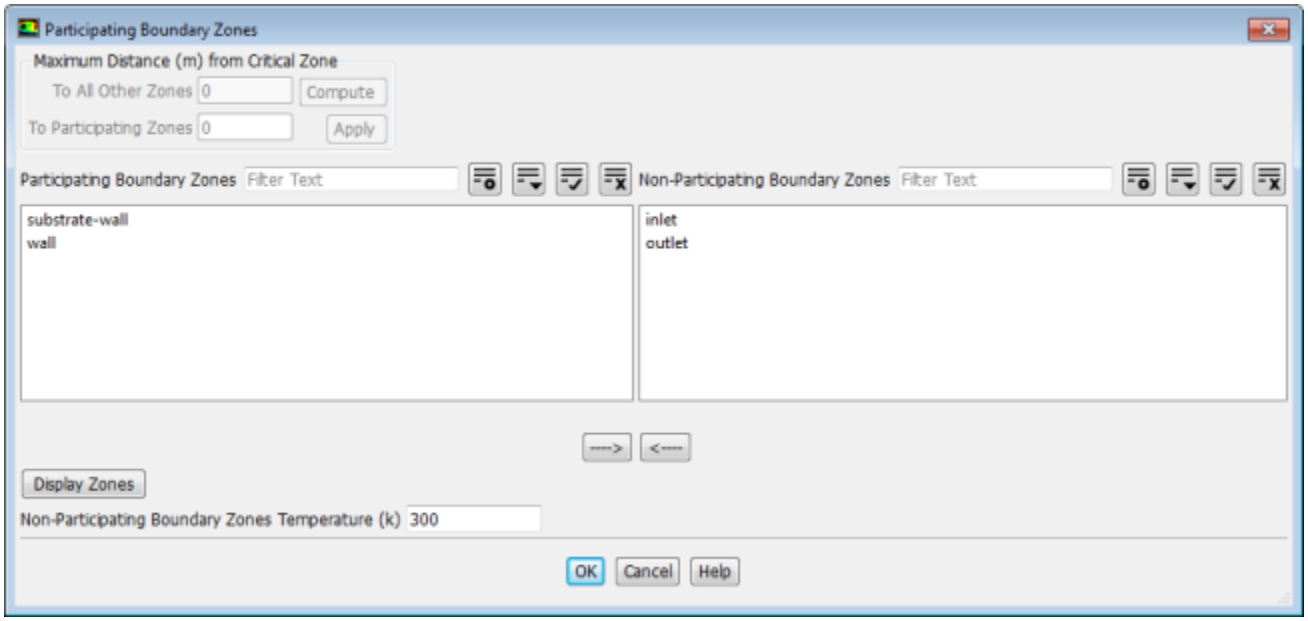
There are two ways in which you can enable/disable the participation of walls and inlet and exit boundaries in the view factor calculation. One of those ways is to use the **Participates in View Factor Calculation** option in the **Radiation** tab of the boundary condition dialog box. The other method is to use the **Participating Boundary Zones** dialog box (Figure 12.17: The Participating Boundary Zones Dialog Box (p. 859)), which is accessed by clicking the **Select...** button next to the **Zones Participating in View Factor Calculation** label in the **View Factors and Clustering** dialog box. For cases that are made up of a very large number of zones, such as underhood applications, the latter method is recommended.

Important

If you compute the view factors and then later alter which boundary zones participate in the view factor calculation, you must recompute the view factors so that the data is up to date.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 12.17: The Participating Boundary Zones Dialog Box



The **Participating Boundary Zones** dialog box allows you to easily specify those zones that are participating or non-participating without having to visit the boundary conditions dialog box of each zone. For cases that have a small number of boundary zones, you can simply select the zones that you do not want to participate in the view factor calculation from the **Participating Boundary Zones** list and click the arrow button that points to the right, so that the zones are moved to the **Non-Participating Boundary Zones** list; if you make an error, you can always reverse this process (that is, select a zone in the **Non-Participating Boundary Zones** list and click the arrow button that points to the left). This can be cumbersome for cases that have a large number of boundary zones, and so the following procedure is recommended instead:

1. Make sure that your clustering options (see Forming Surface Clusters (p. 853)) are appropriate for your view factor settings:

   a. Select **Automatic** from the **Options** list in the **Clustering** group box of the **View Factors and Clustering** dialog box, as this enables some GUI items in the **Participating Boundary Zones** dialog box.

   Important

   Note that if you do not want to use the **Automatic** option for the clustering, you can revert to the **Manual** option after you are done using the **Participating Boundary Zones** dialog box.

   b. Verify that the walls you specified as critical (by enabling the **Critical Zone** option in the **Radiation** tab of the boundary condition dialog box) also correspond to those that are critical for the view factor calculation. You must have specified at least one critical zone for the steps that follow.

2. Click the **Compute** button in the **Maximum Distance from Critical Zone** dialog box, to update the value displayed for **To All Other Zones**. This value represents the maximum distance between the centroids of a critical zone and a non-critical zone in the mesh; it is for information purposes only, and cannot be edited in this dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Based on the value displayed for **To All Other Zones**, enter a threshold value for **To Participating Zones**. The value you enter will specify the maximum distance allowed between the centroids of a critical zone and a zone that participates in the view factor calculation. Then click the **Apply** button to move all zones beyond this distance into the **Non-Participating Boundary Zones** list.

4. Review the zones displayed in the **Participating Boundary Zones** and **Non-Participating Boundary Zones** lists. If necessary, select zones in these lists and use the arrow buttons to move them to the appropriate list, as described previously.

If at any point you want to visually identify zones displayed in the **Participating Boundary Zones** and **Non-Participating Boundary Zones** lists, select the zones and click the **Display Zones** button. Only the selected zones will be displayed in the graphics window.

If any zones are displayed in the **Non-Participating Boundary Zones** list, ensure to enter an appropriate temperature for **Non-Participating Boundary Zones Temperature**. In most cases the appropriate value is the ambient temperature, which by default is assumed to be 300 K.

After you have specified which zones do not participate in view factor calculation and set their temperature, click **OK** to store the settings and close the **Participating Boundary Zones** dialog box. You can then proceed to computing the view factors, as described in the section that follows.

12.3.4.2. Computing View Factors

ANSYS Fluent can compute the view factors for your problem in the current session and save them to a file for use in the current session and future sessions. Alternatively, you can save the surface cluster information and view factor parameters to a file, calculate the view factors outside ANSYS Fluent, and then read the view factors into ANSYS Fluent. These methods for computing view factors are described in the following sections.

Important

For large meshes or complex models, it is recommended that you calculate the view factors outside ANSYS Fluent and then read them into ANSYS Fluent before starting your simulation.

Note

You can accelerate view factor calculations by using ANSYS Fluent in parallel. For more information see Accelerating View Factor Calculations for General Purpose Computing on Graphics Processing Units (GPGPUs) (p. 2064).

12.3.4.2.1. Computing View Factors Inside ANSYS Fluent

To compute view factors in your current ANSYS Fluent session, you must first set the parameters for the view factor calculation in the **View Factors and Clustering** dialog box (see View Factors and Clustering Settings (p. 853) for details) and click **OK** to save them. When you have set the view factor and surface cluster parameters, click the **Compute/Write/Read...** button in the **View Factors and Clustering** group box of the **Radiation Model** dialog box. A **Select File** dialog box will open, prompting you for the name of the file in which ANSYS Fluent should save the surface cluster information and the view factors. Note that the option to **Write Binary Files** is enabled by default, in order to reduce the time needed to write and read the view factor files. When you click **OK** in the **Select File** dialog box, ANSYS Fluent will write the surface cluster information to the file. ANSYS Fluent will use the surface cluster information to compute the view factors, save the view factors to the same file, and then automatically

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

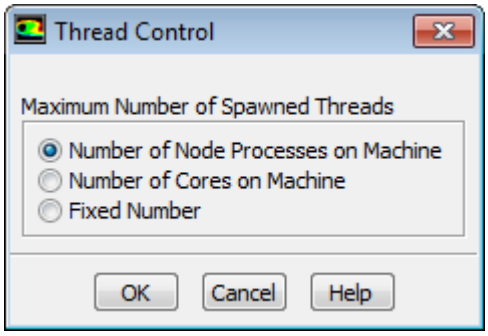
read the view factors. The ANSYS Fluent console will report the status of the view factor calculation. For example:

```
Completed 25% calculation of view factors

Completed 50% calculation of view factors

Completed 75% calculation of view factors

Completed 100 % calculation of view factors
```

Important

- You must recompute the view factors if you take any of following actions after the initial computation:
  - if you alter which boundary zones participate in the view factor calculation (either by using the **Participating Boundary Zones** dialog box, or by enabling / disabling the **Participates in View Factor Calculation** option in the **Radiation** tab of the boundary condition dialog box)
  - if you scale or unscale the mesh by using the **Scale Mesh** dialog box
- The view factor file format for this version of ANSYS Fluent is known as the compressed row format (CRF) and is a more efficient way of writing view factors than in versions that are prior to Fluent 6.4. In the CRF format, only nonzero view factors with their associated cluster IDs are stored to the file. This reduces the size of the `.s2s` file, and reduces the time it takes to read the file into ANSYS Fluent. While the CRF file format is the default, you can still use the older file format if necessary. Contact your support engineer for more information.
- View factors using ray tracing are computed with the MPI/OpenMP hybrid model for parallel runs. By default, one OpenMP thread is used per MPI process, as specified in the Thread Control Dialog Box (p. 2811) (see Controlling the Threads (p. 2058) for details). To use all the available cores on your machine as OpenMP threads, select **Number of Cores on Machine**. A **Fixed Number** of cores can also be used based on your needs.

Figure 12.18: The Thread Control Dialog Box



Note

For cases that use the S2S model and contain non-conformal interfaces, if the mesh interfaces are defined after the case has been read, then the intersection of the interface zones will occur during the initialization process, which will cause relabeling of faces

throughout the domain. If you attempt to compute view factors before solution initialization in such a case, then the following message will be displayed in the console and the view factors will be computed automatically during solution initialization to avoid a mismatch with the case file:

```
Intersection zones for non-conformal interfaces are not created.
View factors will be computed during solution initialization.
```

12.3.4.2.2. Computing View Factors Outside ANSYS Fluent

To compute view factors outside ANSYS Fluent, you must save the surface cluster information and view factor parameters to a file.

File → **Write** → **Surface Clusters...**

ANSYS Fluent will open the **View Factors and Clustering** dialog box, where you will set the view factor and surface cluster parameters (see View Factors and Clustering Settings (p. 853) for details). When you click **OK** in the **View Factors and Clustering** dialog box, a **Select File** dialog box will open, prompting you for the name of the file in which ANSYS Fluent should save the surface cluster information and view factor parameters to the file. If the specified **Filename** ends in `.gz` or `.Z`, appropriate file compression will be performed.

To calculate the view factors outside ANSYS Fluent, enter one of the following commands:

- For the serial solver:

  `utility viewfac` *inputfile*

  where *inputfile* is the filename, or the correct path to the filename, for the surface cluster information and view factor parameters file that you saved from ANSYS Fluent. You can then read the view factors into ANSYS Fluent, as described below.

- For the parallel solver:

  – Viewfac utility module:

  `utility viewfac -t` *nprocs* `[-p` *interconnect* `] [-mpi=` *mpi_type* `] [-cnf=` *hosts* `]` *inputfile*

  where *inputfile* is the filename, for which you will need to provide the full path.

  – Raytracing utility module:

  `utility raytracing -t` *nprocs* `[-p` *interconnect* `] [-mpi=` *mpi_type* `] [-cnf=` *hosts* `] [-ompthreads=` *N* `]` *inputfile*

  where *inputfile* is the filename, for which you will need to provide the full path.

  On Windows:
  `-t`*nprocs* specifies the number of processes to use. When the `-cnf` option is present, the hosts file argument is used to determine which machines to use for the parallel job. For example, if there are 8 machines listed in the hosts file and you want to run a job with 4 processes, set *nprocs* to 4 (that is `-t4`) and ANSYS Fluent will use the first 4 machines listed in the hosts file. Note that this does not apply to the Compute Cluster Server (CCS).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`-p`*interconnect* (optional) specifies the type of interconnect. The ethernet interconnect is used by default if the option is not explicitly specified. See Table 34.2: Supported Interconnects for the Windows Platform (p. 2023), Table 34.3: Available MPIs for Windows Platforms (p. 2023), and Table 34.4: Supported MPIs for Windows Architectures (Per Interconnect) (p. 2023) for more information.
`-mpi=`*mpi_type* (optional) specifies the type of MPI. If the option is not specified, the default MPI for the given interconnect (IBM MPI) will be used (the use of the default MPI is recommended). The available MPIs for Windows are shown in Table 34.3: Available MPIs for Windows Platforms (p. 2023).
`-cnf=`*hosts* (optional) specifies the hosts file, which contains a list of the machines on which you want to run the parallel job. This option functions the same as that in the command for launching parallel Fluent; see Starting Parallel ANSYS Fluent on a Windows System Using Command Line Options (p. 2021) for details.
`-ompthreads=`*N* (optional) specifies that the desired number of OpenMP threads to be used per machine is *N*.

On Linux:
`-t`*nprocs* specifies the number of processes to used. When the `-cnf` option is present, the hosts file argument is used to determine which machines to use for the parallel job. For example, if there are 10 machines listed in the hosts file and you want to run a job with 5 processes, set *nprocs* to 5 (that is `-t5`) and ANSYS Fluent will use the first 5 machines listed in the hosts file.
`-p`*interconnect* (optional) specifies the type of interconnect. The ethernet interconnect is used by default if the option is not explicitly specified. See Table 34.5: Supported Interconnects for Linux Platforms (Per Platform) (p. 2028), Table 34.6: Available MPIs for Linux Platforms (p. 2028), and Table 34.7: Supported MPIs for Linux Architectures (Per Interconnect) (p. 2028) for more information.
`-mpi=`*mpi_type* (optional) specifies the type of MPI. If the option is not specified, the default MPI for the given interconnect will be used (the use of the default MPI is recommended). The available MPIs for Linux are shown in Table 34.6: Available MPIs for Linux Platforms (p. 2028).
`-cnf=`*hosts* (optional) specifies the hosts file, which contains a list of the machines on which you want to run the parallel job. This option functions the same as that in the command for launching parallel Fluent; see Starting Parallel ANSYS Fluent on a Linux System Using Command Line Options (p. 2026) for details.
`-ompthreads=`*N* (optional) specifies that the desired number of OpenMP threads to be used per machine is *N*.

12.3.4.3. Reading View Factors into ANSYS Fluent

If the view factors for your problem have already been computed (either inside or outside ANSYS Fluent) and saved to a file, you can read them into ANSYS Fluent. To read in the view factors, click **Read Existing File...** button in the **View Factors and Clustering** group box of the **Radiation Model** dialog box. A **Select File** dialog box will open where you can specify the name of the file containing the view factors. Alternatively, you can read the view factors file using the **File/Read/View Factors...** ribbon tab item.

Important

While the previous `.s2s` view factor file format can still be read seamlessly into ANSYS Fluent, there is now a more efficient compressed row format (CRF) that can be read into ANSYS Fluent (see the section on Computing View Factors Inside ANSYS Fluent). You can take advantage of the reduced size of the CRF file and therefore the reduced time it takes to read the file into ANSYS Fluent, by converting the existing old file format to the new format (without having to recompute the view factors) using the following command at the command prompt in your working directory:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
utility viewfac -c1 -o new.s2s.gz old.s2s.gz
```

where `new.s2s.gz` is the CRF format to which you want the old file format (`old.s2s.gz`) converted.

12.3.5. Setting Up the DO Model

For information about setting up the DO model, see the following sections:
12.3.5.1. Angular Discretization
12.3.5.2. Defining Non-Gray Radiation for the DO Model
12.3.5.3. Enabling DO/Energy Coupling

For information about accelerating the DO solver when running in parallel on Linux, see Accelerating Discrete Ordinates (DO) Radiation Calculations (p. 2065).

12.3.5.1. Angular Discretization

When you select the **Discrete Ordinates** model, the **Radiation Model** dialog box will expand to show inputs for **Angular Discretization** (see Figure 12.11: The Radiation Model Dialog Box (DO Model) (p. 846)). In this section, you will set parameters for the angular discretization and pixelation described in Angular Discretization and Pixelation in the Theory Guide.

Theta Divisions (N_θ) and **Phi Divisions** (N_ϕ) will define the number of control angles used to discretize each octant of the angular space (see Figure 5.3: Angular Coordinate System in the Theory Guide). Note that higher levels of discretization are recommended for problems where specular exchange of radiation is important to increase the likelihood of the correct beam direction being captured. For a 2D model, ANSYS Fluent will solve only 4 octants (due to symmetry); therefore, a total of $4N_\theta N_\phi$ directions $\vec{s}$ will be solved. For a 3D model, 8 octants are solved, resulting in $8N_\theta N_\phi$ directions $\vec{s}$. By default, the number of **Theta Divisions** and the number of **Phi Divisions** are both set to 2. For most practical problems, these settings are acceptable, however, a setting of 2 is considered to be a coarse estimate. Increasing the discretization of **Theta Divisions** and **Phi Divisions** to a minimum of 3, or up to 5, will achieve more reliable results. A finer angular discretization can be specified to better resolve the influence of small geometric features or strong spatial variations in temperature, but larger numbers of **Theta Divisions** and **Phi Divisions** will add to the cost of the computation.

Theta Pixels and **Phi Pixels** are used to control the pixelation that accounts for any control volume overhang (see Figure 5.7: Pixelation of Control Angle in the Theory Guide and the figures and discussion preceding it). For problems involving gray-diffuse radiation, the default pixelation of 1×1 is usually sufficient. For problems involving symmetry, periodic, specular, or semi-transparent boundaries, a pixelation of 3×3 is recommended and will achieve acceptable results. The computational effort, as a result of increasing the pixelation, is less than the computational effort caused by increasing the divisions. You should be aware, however, that increasing the pixelation adds to the cost of computation.

Important

Note that pixelations are applied to boundary faces by default.

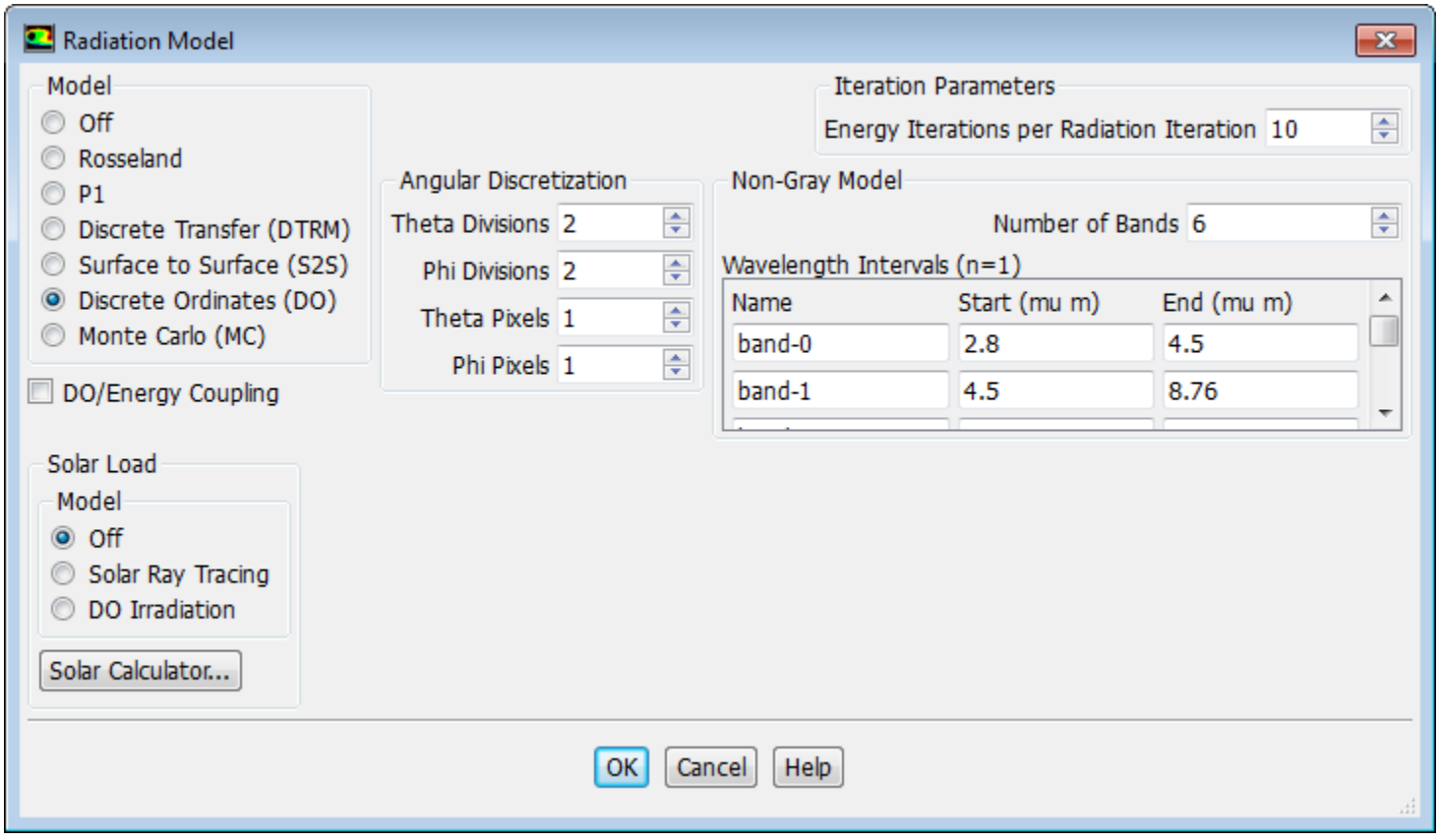
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12.3.5.2. Defining Non-Gray Radiation for the DO Model

If you want to model non-gray radiation using the DO model, you can specify the **Number of Bands** (N) under **Non-Gray Model** in the expanded **Radiation Model** dialog box (Figure 12.19: The Radiation Model Dialog Box (Non-Gray DO Model) (p. 865)). For a 2D model, ANSYS Fluent will solve $4N_\theta N_\phi N$ directions. For a 3D model, $8N_\theta N_\phi N$ directions will be solved. By default, the **Number of Bands** is set to zero, indicating that only gray radiation will be modeled. Because the cost of computation increases directly with the number of bands, you should try to minimize the number of bands used. In many cases, the absorption coefficient or the wall emissivity is effectively constant for the wavelengths of importance in the temperature range of the problem. For such cases, the gray DO model can be used with little loss of accuracy. For other cases, non-gray behavior is important, but relatively few bands are necessary. For typical glasses, for example, two or three bands will frequently suffice.

When a nonzero **Number of Bands** is specified, the **Radiation Model** dialog box will expand once again to show the **Wavelength Intervals** (Figure 12.19: The Radiation Model Dialog Box (Non-Gray DO Model) (p. 865)). You can specify a **Name** for each wavelength band, as well as the **Start** and **End** wavelength of the band in μm. Note that the wavelength bands are specified for vacuum (n=1). ANSYS Fluent will automatically account for the refractive index in setting band limits for media with n different from unity.

Figure 12.19: The Radiation Model Dialog Box (Non-Gray DO Model)



The frequency of radiation remains constant as radiation travels across a semi-transparent interface. The wavelength, however, changes such that $n\lambda$ is constant. Therefore, when radiation passes from a medium with refractive index n_1 to one with refractive index n_2, the following relationship holds:

$$n_1\lambda_1 = n_2\lambda_2 \tag{12.8}$$

Here λ_1 and λ_2 are the wavelengths associated with the two media. It is conventional to specify the wavelength rather than frequency. ANSYS Fluent requires you to specify wavelength bands (in μm) for an equivalent medium with $n=1$.

For example, consider a typical glass with a step jump in the absorption coefficient at a cut-off wavelength of λ_c. The absorption coefficient is a_1 for $\lambda \leq \lambda_c$ and a_2 for $\lambda > \lambda_c$. The refractive index of the glass is n_g. Since $n\lambda$ is constant across a semi-transparent interface, the equivalent cut-off wavelength for a medium with $n=1$ is $n_g\lambda_c$ using Equation 12.8 (p. 865). You should choose two bands in this case, with the limits 0 to $n_g\lambda_c$ and $n_g\lambda_c$ to 100. Here, the upper wavelength limit has been chosen to be a large number, 100, in order to ensure that the entire spectrum is covered by the bands. When multiple materials exist, you should convert all the cut-off wavelengths to equivalent cut-off wavelengths for an $n=1$ medium, and choose the band boundaries accordingly.

The bands can have different widths and need not be contiguous. You can ensure that the entire spectrum is covered by your bands by choosing $\lambda_{min}=0$ and $n\lambda_{max}T_{min} \geq 50,000$. Here λ_{min} and λ_{max} are the minimum and maximum wavelength bounds of your wavelength bands, and T_{min} is the minimum expected temperature in the domain.

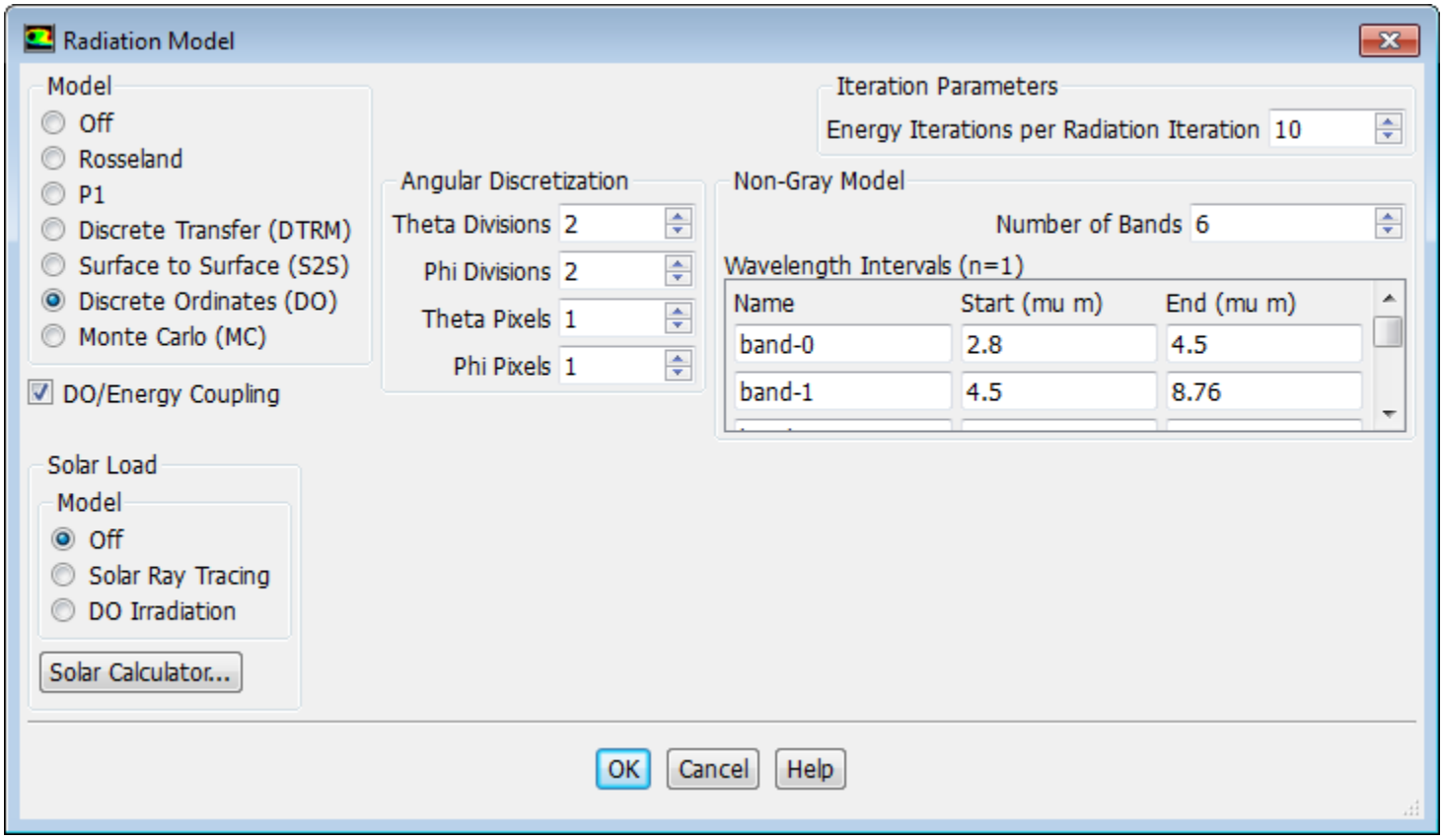
ANSYS Fluent allows you to use a user-defined function (UDF) to modify the emissivity weighting factor $F(0 \rightarrow n\lambda_2 T) - F(0 \rightarrow n\lambda_1 T)$ (which otherwise defaults to the black body emission factor obtained from a standard Planck distribution). The emissivity weighting factor appears in the emission term of the radiative transfer equation for the non-gray model, as shown in Equation 5.61 in the Theory Guide. For more information, see `DEFINE_EMISSIVITY_WEIGHTING_FACTOR` in the Fluent Customization Manual.

12.3.5.3. Enabling DO/Energy Coupling

For applications involving optical thicknesses greater than 10, you can enable the **DO/Energy Coupling** option in the **Radiation Model** (Figure 12.20: The Radiation Model Dialog Box with DO/Energy Coupling Enabled (p. 867)) in order to couple the energy and intensity equations at each cell, solving them simultaneously. This approach accelerates the convergence of the finite volume scheme for radiative heat transfer and can be used with the gray or non-gray radiation model.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 12.20: The Radiation Model Dialog Box with DO/Energy Coupling Enabled



12.3.6. Setting Up the MC Model

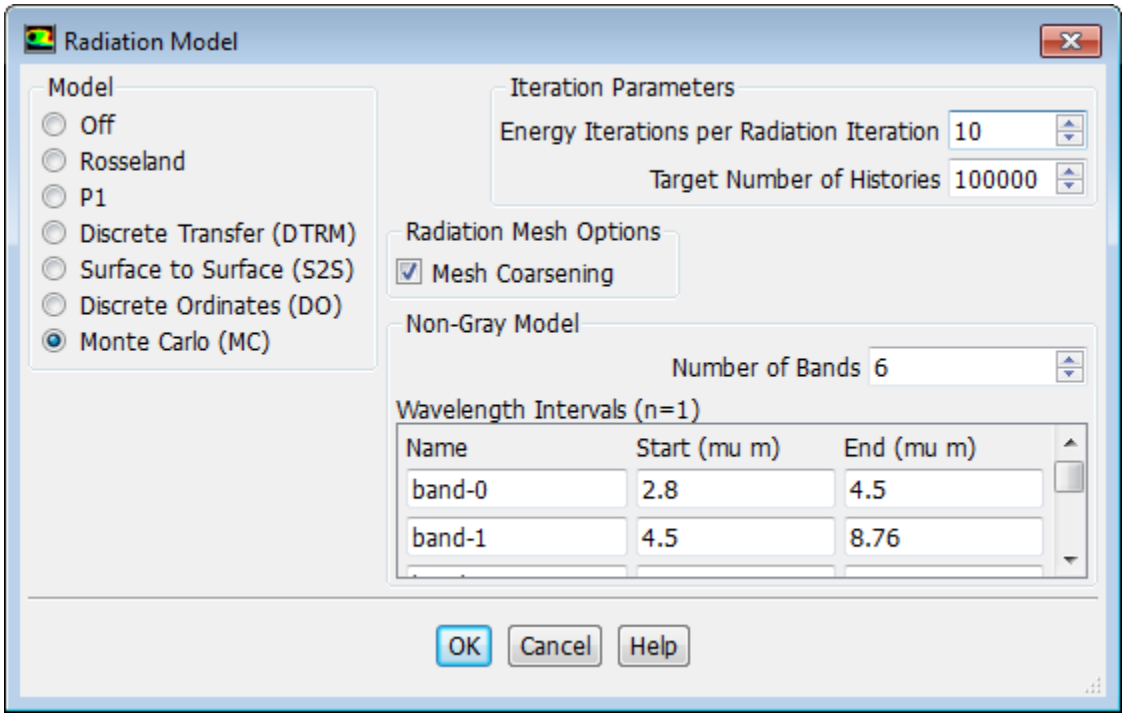
In the **Iteration Parameters** group box of the **Radiation Model** dialog box, you have the option of revising the **Target Number of Histories** to be tracked for the Monte Carlo (MC) simulation. The default value is 100,000. This can have a large effect on the accuracy of simulation results. For details, see Monte Carlo Solution Accuracy in the Theory Guide.

In the **Radiation Mesh Options** group box, you can enable **Mesh Coarsening**. **Mesh Coarsening** reduces the number of effective cells in the domain used by the MC radiation model. The coarsening is achieved by merging cells together and, therefore, does not affect the surface mesh resolution or alter the faces of any merged cells. The reduction in cells may speed up the radiation calculation and reduce peak memory usage, however, it may also have an impact on solution accuracy.

You have the option specifying multiple radiation bands using the **Non-Gray Model** group box. For each desired band, you must specify a start and end wavelength.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 12.21: The Radiation Model Dialog Box (MC)



12.3.7. Defining Material Properties for Radiation

When you are using the P-1, DO, MC, or Rosseland radiation model in ANSYS Fluent, you should be sure to define both the absorption and scattering coefficients of the fluid in the **Create/Edit Materials** dialog box. Note that you can either enter a constant value for these parameters, or you can specify them using a user-defined function (UDF). For more information, see DEFINE_PROPERTY UDFs in the Fluent Customization Manual.

Setup → Materials

If you are modeling semi-transparent media using the DO or MC model, you should also define the refractive index for the semi-transparent fluid or solid material. When using the Rosseland model, you can specify the refractive index only for the fluid material. When using the P-1 model, you should define the refractive index for the fluid material only. For the DTRM, you need to define only the absorption coefficient.

If your model includes gas phase species such as combustion products, absorption and/or scattering in the gas may be significant. The scattering coefficient should be increased from the default of zero if the fluid contains dispersed particles or droplets that contribute to scattering. Alternatively, you can specify the scattering coefficient as a user-defined function (UDF). For more information, see DEFINE_PROPERTY UDFs in the Fluent Customization Manual.

Note

You cannot use a UDF to define any material properties when using the MC model.

ANSYS Fluent allows you to enter a composition-dependent absorption coefficient for CO_2 and H_2O mixtures, using the WSGGM. The method for computing a variable absorption coefficient is described

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in Radiation in Combusting Flows in the Theory Guide. Radiation Properties (p. 515) provides a detailed description of the procedures used for specification of radiation properties.

12.3.7.1. Absorption Coefficient for a Non-Gray Model

If you are using the non-gray P-1, DO, or MC model, you can specify a different constant absorption coefficient for each of the bands used by the gray-band model, as described in Radiation Properties (p. 515). You cannot, however, compute a composition-dependent absorption coefficient in each band. If you use the WSGGM to compute a variable absorption coefficient, the value will be the same for all bands. Alternatively, you can specify a user-defined function (UDF) for the absorption coefficient. For more information, see DEFINE_PROPERTY UDFs in the Fluent Customization Manual.

12.3.7.2. Refractive Index for a Non-Gray Model

If you are using the non-gray P-1, DO, or MC model, you can specify a different constant refractive index for each of the bands used by the gray-band model, as described in Radiation Properties (p. 515). You cannot, however, compute a composition-dependent refractive index in each band.

12.3.8. Defining Boundary Conditions for Radiation

When you set up a problem that includes radiation, you will set additional boundary conditions at inlets, exits, and walls. These inputs are described in the sections that follow. The boundary condition dialog boxes can be opened by right-clicking the boundary name in the tree (under **Setup/Boundary Conditions**) and clicking **Edit...** in the menu that opens; alternatively, you can open them from the **Boundary Conditions** task page:

Setup → **Boundary Conditions**

12.3.8.1. Inlet and Exit Boundary Conditions

12.3.8.1.1. Emissivity

When radiation is active, you can define the emissivity at each inlet and exit boundary when you are defining boundary conditions in the associated inlet or exit boundary dialog box (for example, **Pressure Inlet** dialog box, **Velocity Inlet** dialog box, **Pressure Outlet** dialog box). Enter the appropriate value for **Internal Emissivity**. The default value for all boundary types is 1. Alternatively, you can specify a user-defined function for emissivity. For more information, see DEFINE_PROFILE in the Fluent Customization Manual.

For the non-gray P-1 model and the non-gray DO model, the specified constant emissivity will be used for all wavelength bands.

Important

The **Internal Emissivity** boundary condition is not available with the Rosseland model.

12.3.8.1.2. Black Body Temperature

ANSYS Fluent includes an option that allows you to take into account the influence of the temperature of the gas and the walls beyond the inlet/exit boundaries, and specify different temperatures for radiation and convection at inlets and exits. This is useful when the temperature outside the inlet or exit differs considerably from the temperature in the enclosure. For example, if the temperature of the walls beyond

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the inlet is 2000 K and the temperature at the inlet is 1000 K, you can specify the outside wall temperature to be used for computing radiative heat flux, while the actual temperature at the inlet is used for calculating convective heat transfer. To do this, you would specify a radiation temperature of 2000 K as the black body temperature.

Although this option allows you to account for both cooler and hotter outside walls, you must use caution in the case of cooler walls, since the radiation from the immediate vicinity of the hotter inlet or outlet almost always dominates over the radiation from cooler outside walls. If, for example, the temperature of the outside walls is 250 K and the inlet temperature is 1500 K, it might be misleading to use 250 K for the radiation boundary temperature. This temperature might be expected to be somewhere between 250 K and 1500 K; in most cases it will be close to 1500 K. Its value depends on the geometry of the outside walls and the optical thickness of the gas in the vicinity of the inlet.

In the flow inlet or exit dialog box (for example, **Pressure Inlet** dialog box, **Velocity Inlet** dialog box), select **Specified External Temperature** in the **External Black Body Temperature Method** drop-down list, and then enter the value of the radiation boundary temperature as the **Black Body Temperature**.

Important

- If you want to use the same temperature for radiation and convection, retain the default selection of **Boundary Temperature** as the **External Black Body Temperature Method**.
- The **Black Body Temperature** boundary condition is not available with the Rosseland model.

12.3.8.2. Wall Boundary Conditions for the DTRM, and the P-1, S2S, and Rosseland Models

The DTRM and the S2S, Rosseland, and gray P-1 (that is, **Number of Bands** is set to zero) models assume all walls to be gray and diffuse. The only radiation boundary condition required in the **Wall** dialog box is the emissivity. For the Rosseland model, the internal emissivity is 1. For the DTRM and the S2S and gray P-1 models, you can enter the appropriate value for **Internal Emissivity** in the **Thermal** tab of the **Wall** dialog box. The default value is 1. Alternatively, you can specify a user-defined function for emissivity. For more information, see `DEFINE_PROFILE` in the Fluent Customization Manual.

For the non-gray P–1 model, specify a constant **Internal Emissivity** for each wavelength band in the **Radiation** tab of the **Wall** dialog box (the default value in each band is 1). Alternatively, you can specify the internal emissivity using a boundary condition parameter (see Creating a New Parameter (p. 246)). See Defining Boundary Conditions for Radiation (p. 869) for details.

12.3.8.2.1. Boundary Conditions for the S2S Model

When the S2S model is used, you can specify that some of the walls and inlet and exit boundaries are not participating in the view factor calculation. This capability allows you to save time computing the view factors and also reduce the memory required to store the view factor file during the ANSYS Fluent calculation. See Specifying Boundary Zone Participation (p. 858) for details.

Important

- Whenever you revise which boundary zones participate in the calculation, you will need to re-compute the view factors.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The **Flux Reports** dialog box will not show the exact balance of the **Radiation Heat Transfer Rate** because the radiative heat transfer to those boundaries that do not participate in the view factor calculation is not included.

12.3.8.3. Wall Boundary Conditions for the DO Model

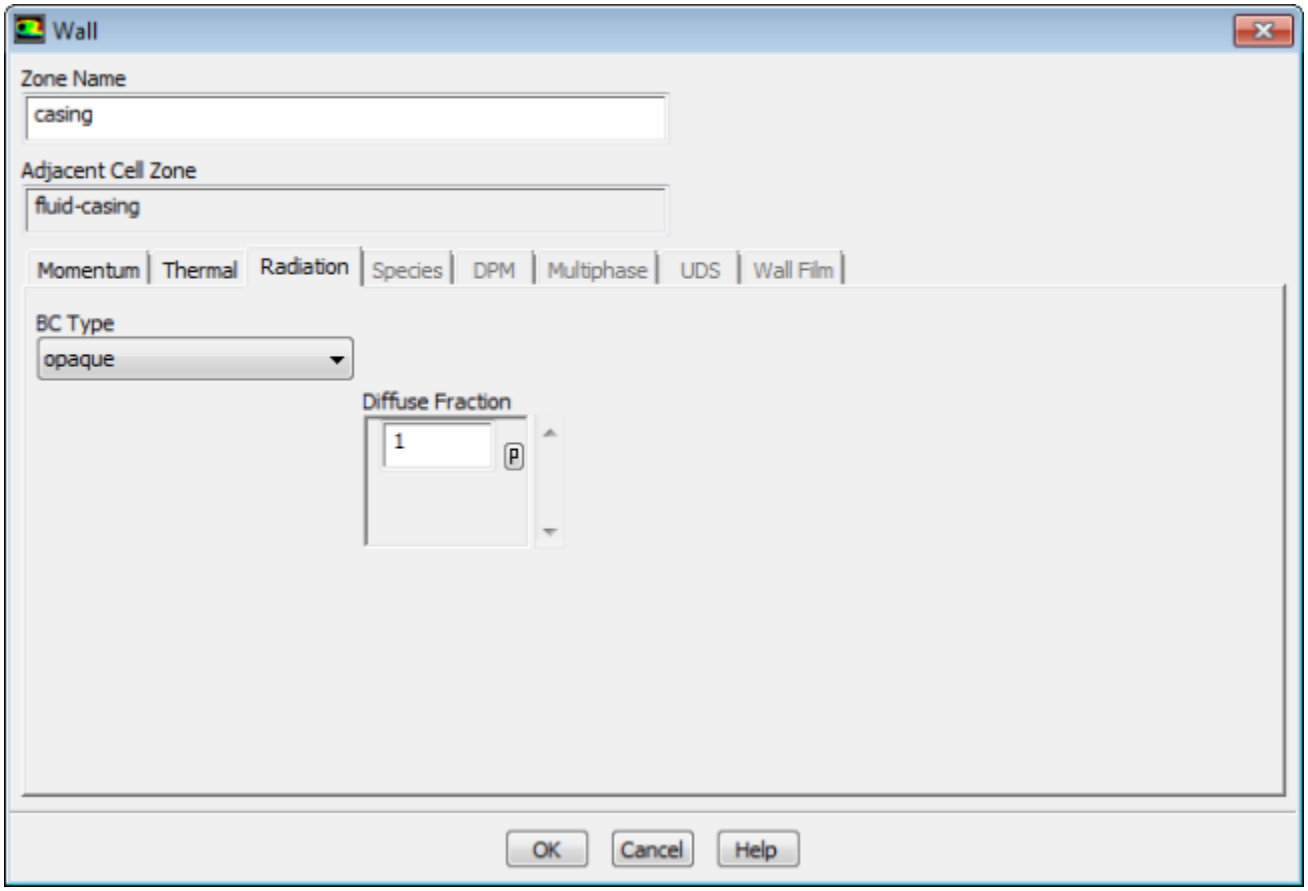
When the DO model is used, you can model opaque walls, as discussed in Boundary and Cell Zone Condition Treatment at Opaque Walls in the Theory Guide, as well as semi-transparent walls (Cell Zone and Boundary Condition Treatment at Semi-Transparent Walls in the Theory Guide).

You can use a diffuse wall to model wall boundaries in many industrial applications since, for the most part, surface roughness makes the reflection of incident radiation diffuse. For highly polished surfaces, such as reflectors or mirrors, the specular boundary condition is appropriate. The semi-transparent boundary condition can be appropriate, for example, when modeling for glass panes in air.

12.3.8.3.1. Opaque Walls

In the **Radiation** tab of the **Wall** dialog box (Figure 12.22: The Wall Dialog Box Showing Radiation Conditions for an Opaque Wall (p. 871)), select **opaque** in the **BC Type** drop-down list to specify an opaque wall. Opaque walls are treated as gray if gray radiation is being computed, or non-gray if the non-gray DO model is being used. If the non-gray DO model is being used, the **Diffuse Fraction** can be specified for each band.

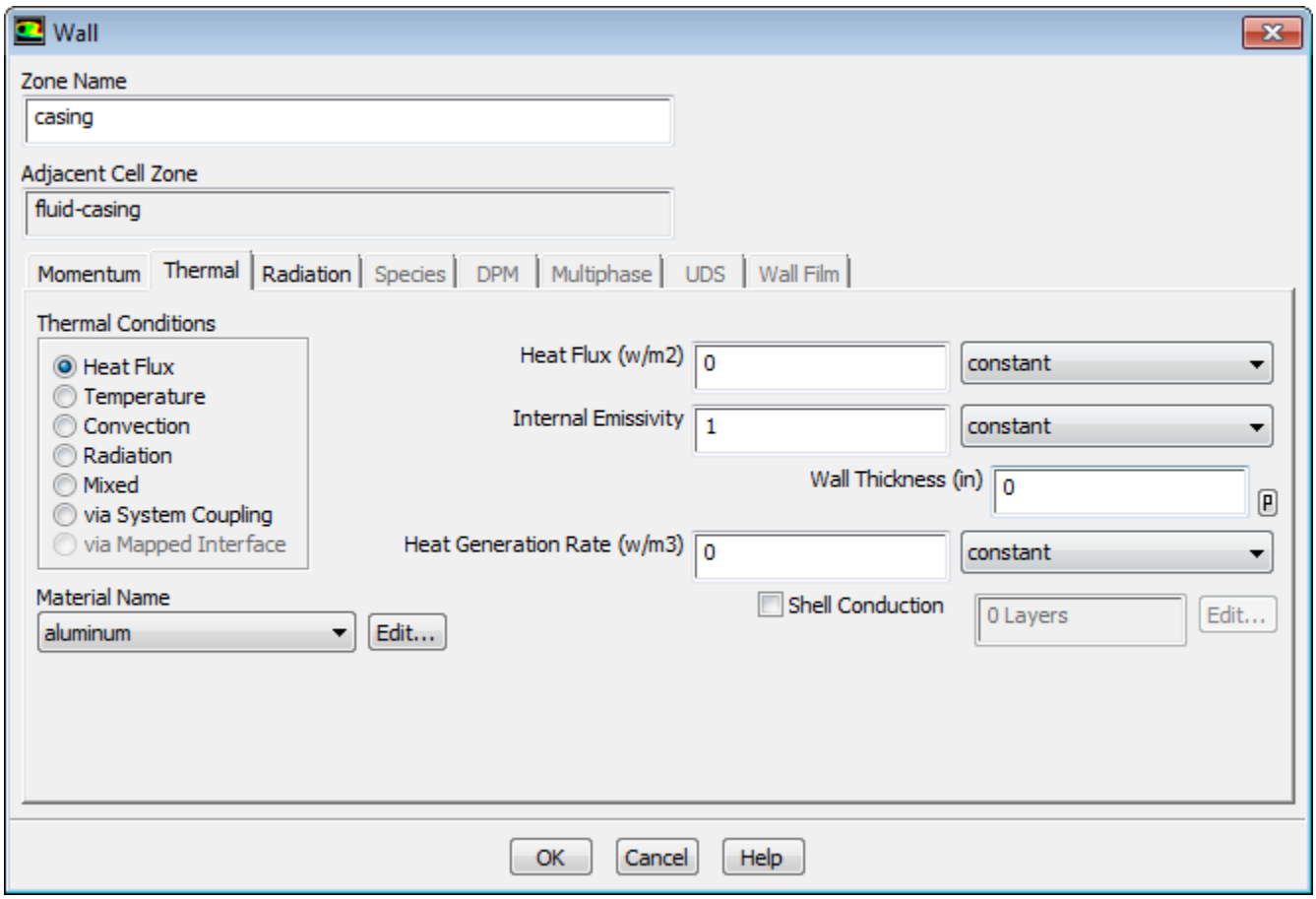
Figure 12.22: The Wall Dialog Box Showing Radiation Conditions for an Opaque Wall



After you have selected **opaque** as the **BC Type**, you can specify the fraction of reflected radiation flux that is to be treated as diffuse. By default, the **Diffuse Fraction** is set to 1, indicating that all of the radiation is diffuse. A diffuse fraction equal to 0 indicates purely specular reflected radiation. A diffuse fraction between 0 and 1 will result in partially diffuse and partially specular reflected energy. See Boundary and Cell Zone Condition Treatment at Opaque Walls in the Theory Guide for more details.

You will also be required to specify the internal emissivity in the **Thermal** tab of the **Wall** dialog box (Figure 12.23: The Wall Dialog Box Showing Internal Emissivity Thermal Conditions for an Opaque Wall (p. 872)). For gray-radiation DO models, enter the appropriate value for **Internal Emissivity**. (The default value is 1.) The value that you specify will be applied to the diffuse component only. For non-gray DO models, specify a constant **Internal Emissivity** for each wavelength band in the **Radiation** tab of the **Wall** dialog box. (The default value in each band is 1.) Alternatively, you can specify the internal emissivity using a boundary condition parameter (see Creating a New Parameter (p. 246)).

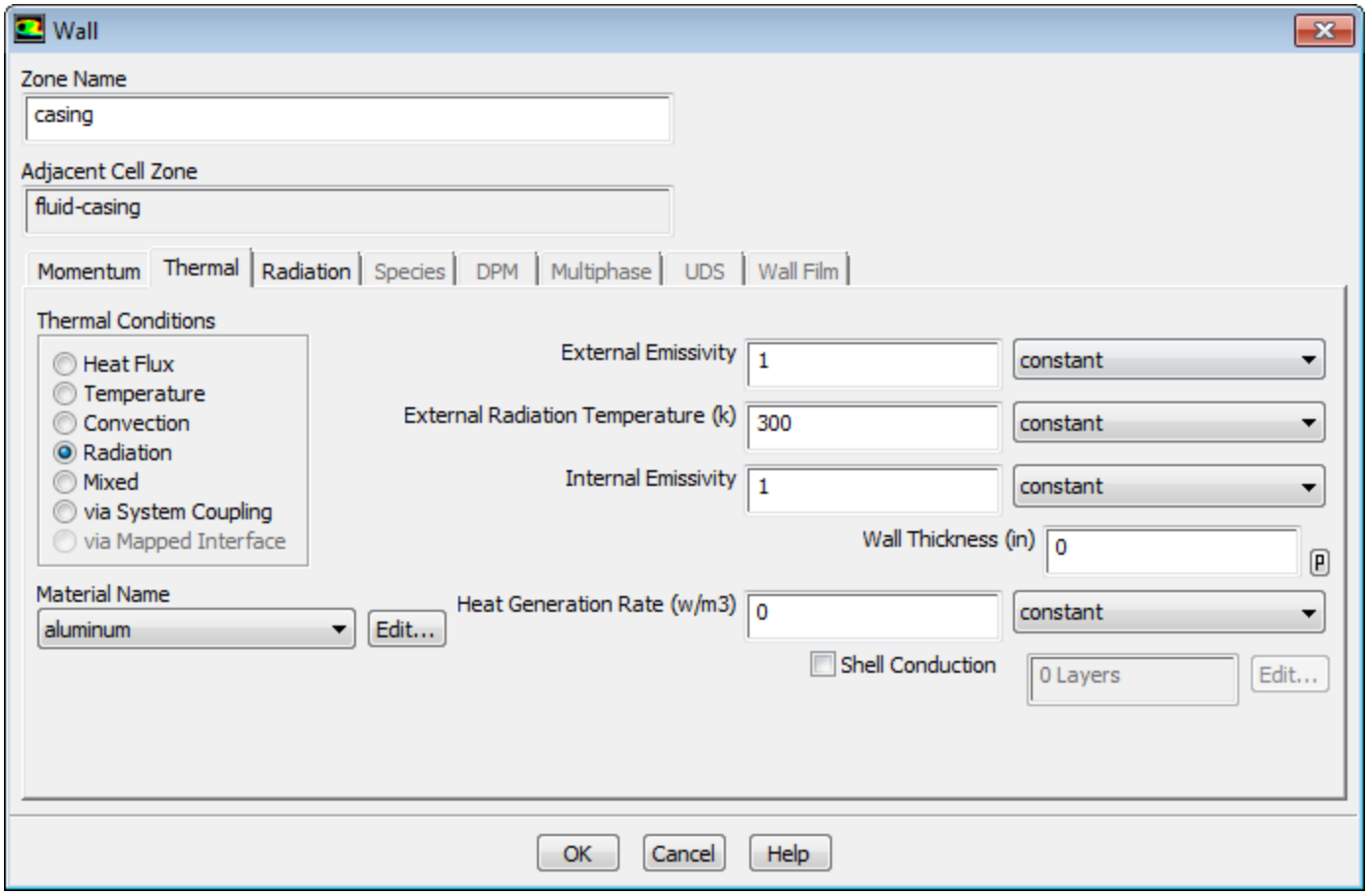
Figure 12.23: The Wall Dialog Box Showing Internal Emissivity Thermal Conditions for an Opaque Wall



You can also specify the external emissivity and external radiation temperature for an opaque wall when the thermal conditions are set to **Radiation** or **Mixed** in the **Wall** dialog box (Figure 12.24: The Wall Dialog Box Showing External Emissivity and External Radiation Temperature Thermal Conditions (p. 873)). Alternatively, you can specify a UDF for these parameters; for more information, see `DEFINE_PROFILE` in the Fluent Customization Manual.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 12.24: The Wall Dialog Box Showing External Emissivity and External Radiation Temperature Thermal Conditions



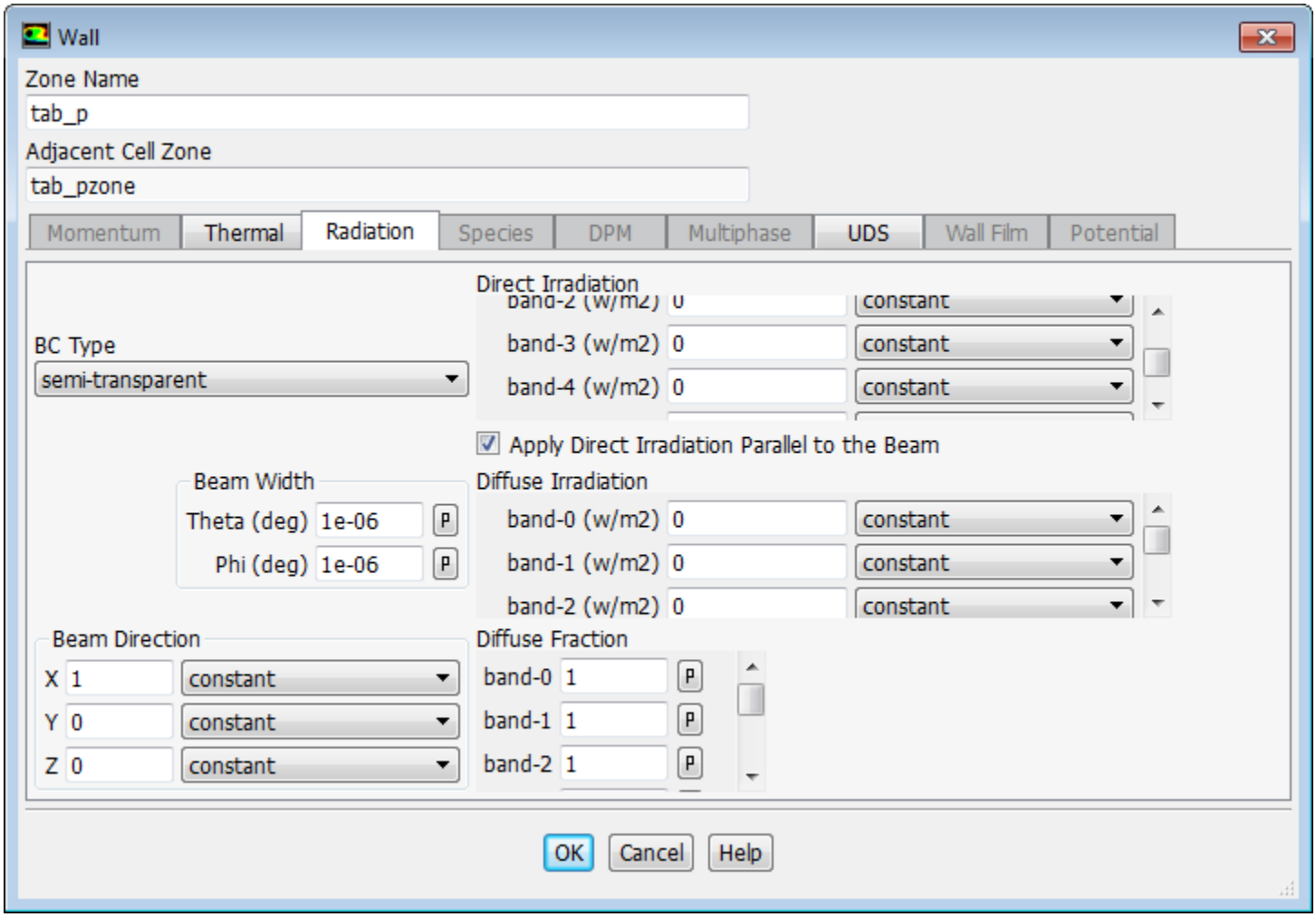
For more information on boundary condition treatment at opaque walls, see Boundary and Cell Zone Condition Treatment at Opaque Walls in the Theory Guide.

12.3.8.3.2. Semi-Transparent Walls

To define radiation for an exterior semi-transparent wall, click the **Radiation** tab in the **Wall** dialog box and then select **semi-transparent** in the **BC Type** drop-down list (Figure 12.25: The Wall Dialog Box for a Semi-Transparent Wall Boundary (p. 874)). The dialog box will expand to display the semi-transparent wall inputs needed to define an external irradiation flux (Figure 12.25: The Wall Dialog Box for a Semi-Transparent Wall Boundary (p. 874)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 12.25: The Wall Dialog Box for a Semi-Transparent Wall Boundary



Then perform the following steps:

1. Specify the value of the irradiation flux (in W/m^2) under **Direct** or **Diffuse Irradiation**. If the non-gray DO model is being used, a constant **Direct** or **Diffuse Irradiation** can be specified for each band.

   Important

   Note that the external diffuse irradiation specified when using **Radiation** or **Mixed** thermal conditions (selected in the **Thermal** tab), or **Diffuse Irradiation** (in the **Radiation** tab) is always distributed hemispherically after transmission through semi-transparent walls (that is independent of whether the external semi-transparent boundary wall is defined as a diffusely or specularly reflecting type).

2. **Apply Direct Irradiation Parallel to the Beam** is the default means of specifying the scale of irradiation flux. When enabled, ANSYS Fluent assumes that the value of **Direct Irradiation** that you specify is the irradiation flux parallel to the **Beam Direction**. When deselected, ANSYS Fluent instead assumes that the value specified is the flux parallel to the face normals and will calculate the resulting beam parallel flux for every face. See Figure 5.12: DO Irradiation on External Semi-Transparent Wall in Semi-Transparent Exterior Walls in the Theory Guide for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Define the **Beam Width** by specifying the beam **Theta** and **Phi** extents. Beam width is specified as the solid angle over which the irradiation is distributed. The default value for beam width is $1e^{-6}$, which is suitable for collimated beam radiation. A beam width less than this is likely to result in zero irradiation flux.

4. Specify the (**X,Y,Z**) vector that defines the **Beam Direction**. The beam direction is defined as the vector of the centroid of the solid angle (beam width). You can specify the **Beam Direction** as a constant, a profile or a UDF. This is especially useful in applications where the shape of the radiative source is circular or cylindrical (or nonlinear). For information about boundary profiles, see Reading and Writing Profile Files (p. 53).

   Note that the actual direction of the beam of radiation that enters the domain will be further influenced by the solid angles available from the number of divisions set up; the effective direction will be the direction vector of the solid angle that the incoming beam falls into. Finally, any nonzero diffuse fraction will act to spread out (hemispherically, proportional to the diffuse fraction) the irradiation that enters the domain.

   For a UDF example that specifies the beam direction, see Example 5 - Beam Direction Profile at Semi-Transparent Walls in the Fluent Customization Manual.

5. Specify the fraction of the irradiation that is to be treated as diffuse as a real number between 0 and 1. By default, the **Diffuse Fraction** is set to 1, indicating that all of the irradiation is diffuse. A diffuse fraction of 0 treats the radiation as purely specular. If you specify a value between 0 and 1, the radiation is treated as partially diffuse and partially specular. If the non-gray DO model is being used, the **Diffuse Fraction** can be specified for each band. See Diffuse Semi-Transparent Walls in the Theory Guide for details.

Important

- Note that the refractive index of the external medium is assumed to be 1.
- If **Heat Flux** conditions are specified in the **Thermal** tab of the **Wall** dialog box, the specified heat flux is considered to be only the conduction and convection portion of the boundary flux. The given irradiation specifies the incoming exterior radiative flux; the radiative flux transmitted from the domain interior to the outside is computed as a part of the calculation by ANSYS Fluent. Internal emissivity is ignored for semi-transparent surfaces.
- Note that when a boundary wall is made semi-transparent ANSYS Fluent calculates the amount of radiation leaving as well as entering the domain. If you do not provide a source of irradiation or a radiating thermal condition (for example **Mixed** or **Radiation**) then you are effectively radiating to a temperature of 0 K and it is highly likely you may observe temperatures in your model that are lower than expected. Ensure that the external (incoming) radiant conditions give good account of the surroundings.

You can also specify the external emissivity and external radiation temperature for a semi-transparent wall when the thermal conditions are set to **Radiation** or **Mixed** in the **Wall** dialog box (Figure 12.24: The Wall Dialog Box Showing External Emissivity and External Radiation Temperature Thermal Conditions (p. 873)). Alternatively, you can specify a user-defined function (UDF) for these parameters; for more information, see `DEFINE_PROFILE` in the Fluent Customization Manual.

For a detailed description of boundary condition treatment at semi-transparent walls, see Cell Zone and Boundary Condition Treatment at Semi-Transparent Walls in the Theory Guide.

To define radiation for an interior (two-sided) semi-transparent wall, in the **Wall** dialog box click the **Radiation** tab and then select **semi-transparent** in the **BC Type** drop-down list (Figure 12.26: The Wall

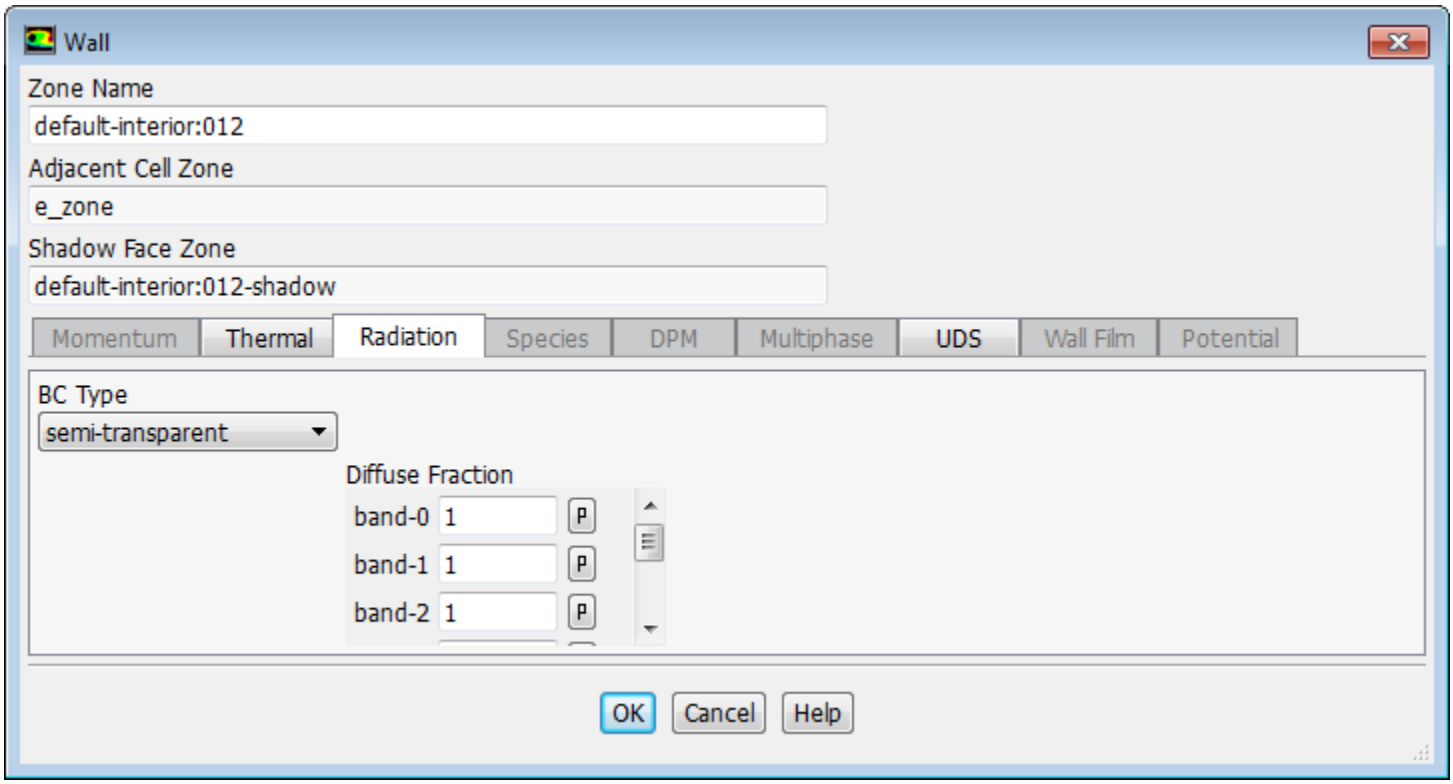
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Dialog Box for an Interior Semi-Transparent Wall (p. 876)). Then specify the **Diffuse Fraction** as described for the previous case.

Important

Note that for semi-transparent walls, the internal emissivity defined under thermal conditions is ignored. If you want to include the effect of internal emissivity at semi-transparent walls, you must create a solid zone and specify that it participates in the radiation calculation; see Solid Semi-Transparent Media in the Theory Guide for details.

Figure 12.26: The Wall Dialog Box for an Interior Semi-Transparent Wall



12.3.8.4. Boundary Conditions for the MC Model

When the MC model is used, you can define the boundaries in a similar manner as for the DO model, with the following differences:

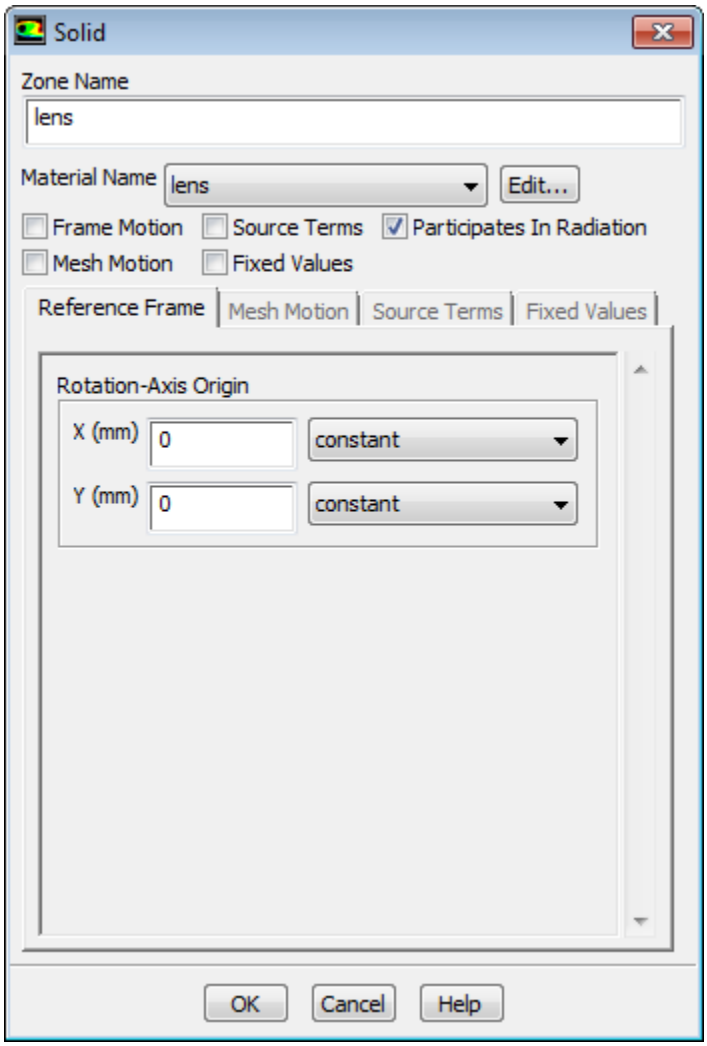
- The MC model does not require a beam width.
- For opaque walls, the MC model allows you to enable a **Boundary Source** option, which allows you to specify the **Beam Direction** and the **Direct Irradiation**.
- To model diffuse irradiation, you must set the **Beam Direction** to 0.
- The semi-transparent boundary condition is not supported on external walls
- Unlike the DO model, direct irradiation cannot be defined for semi-transparent walls.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12.3.8.5. Solid Cell Zones Conditions for the DO Model

With the DO model, you can specify whether or not you want to solve for radiation in each cell zone in the domain. By default, the DO equations are solved in all fluid zones, but not in any solid zones. If you want to model semi-transparent media, for example, you can enable radiation in the solid zone(s). To do so, enable the **Participates In Radiation** option in the **Solid** dialog box (Figure 12.27: The Solid Dialog Box (p. 877)).

Figure 12.27: The Solid Dialog Box



Important

In general, you should *not* disable the **Participates In Radiation** option for any fluid zones.

See Solid Semi-Transparent Media in the Theory Guide for more information on solid semi-transparent media.

12.3.8.6. Thermal Boundary Conditions

In general, any well-posed combination of thermal boundary conditions can be used when any of the radiation models is active. The radiation model will be well-posed in combination with fixed temperature walls, conducting walls, and/or walls with set external heat transfer boundary conditions (Thermal

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Boundary Conditions at Walls (p. 370)). You can also use any of the radiation models with heat flux boundary conditions defined at walls, in which case the heat flux you define will be treated as the sum of the convective and radiative heat fluxes. The exception to this is the case of semi-transparent walls for the DO model. Here, ANSYS Fluent allows you to specify the convective and radiative portions of the heat flux separately.

12.3.9. Solution Strategies for Radiation Modeling

For the P-1, DTRM, S2S, and the DO radiation models, there are several parameters that control the radiation calculation. You can use the default solution parameters for most problems, or you can modify these parameters to control the convergence and accuracy of the solution. Iteration parameters that are unique for a particular radiation model are specified in the **Radiation Model** dialog box (for example, **Energy Iterations per Radiation Iteration**). **Spatial Discretization** (Discretization in the Theory Guide) and **Under-Relaxation Factors** (Under-Relaxation of Variables in the Theory Guide) are specified in the **Solution Methods** and **Solution Controls** task pages, respectively. The **Convergence Criterion** (Modifying Convergence Criteria (p. 1631)) is set in the **Residual Monitors** dialog box.

There are no solution parameters to be set for the Rosseland model, since it impacts the solution only through the energy equation.

Important

If radiation is the only model being solved in ANSYS Fluent and all other equations are disabled, then the radiation transfer equation will automatically be solved for every iteration, regardless of what the **Energy Iterations per Radiation Iteration** field is set to in the user interface.

12.3.9.1. P-1 Model Solution Parameters

For the P-1 radiation model, you can control the convergence criterion and under-relaxation factor. You should also pay attention to the optical thickness, as described below.

The default convergence criterion for the P-1 model is 10^{-6}, the same as that for the energy equation, since the two are closely linked. See Monitoring Residuals (p. 1624) for details about convergence criteria. You can set the **Convergence Criterion** for **p1** in the **Residual Monitors** dialog box.

Solution → **Monitors** → **Residuals** **Edit...**

The under-relaxation factor for the P-1 model is set with those for other variables, as described in Setting Under-Relaxation Factors (p. 1564). Note that since the equation for the radiation temperature (Equation 5.21 in the Theory Guide) is a relatively stable scalar transport equation, in most cases you can safely use large values of under-relaxation (0.9–1.0).

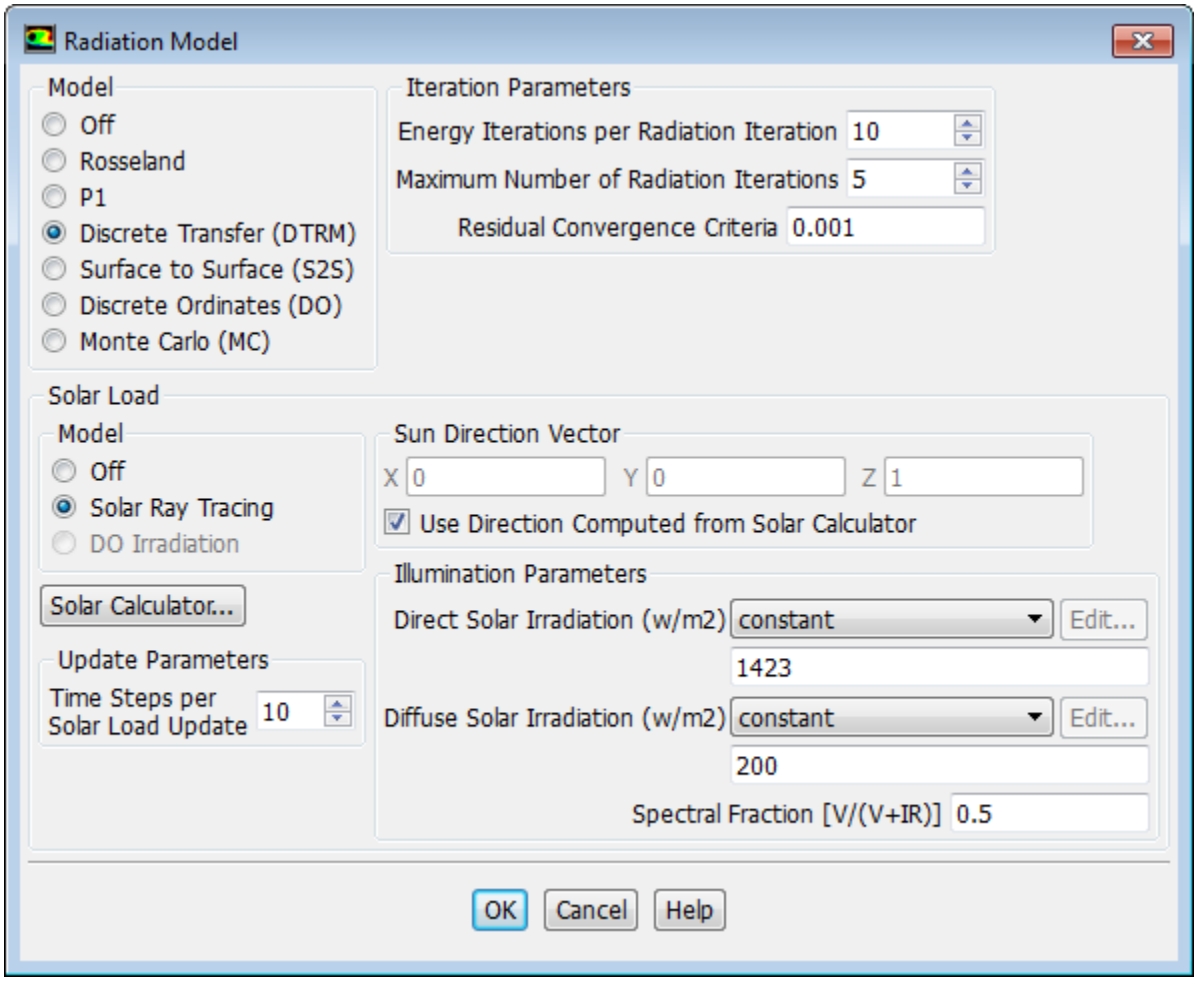
For optimal convergence with the P-1 model, the optical thickness $(a+\sigma_s)L$ must be between 0.01 and 10 (preferably not larger than 5). Smaller optical thicknesses are typical for very small enclosures (characteristic size of the order of 1 cm), but for such problems you can safely increase the absorption coefficient to a value for which $(a+\sigma_s)L=0.01$. Increasing the absorption coefficient will not change the physics of the problem because the difference in the level of transparency of a medium with optical thickness = 0.01 and one with optical thickness <0.01 is indistinguishable within the accuracy level of the computation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12.3.9.2. DTRM Solution Parameters

When the DTRM is active, ANSYS Fluent updates the radiation field during the calculation and computes the resulting energy sources and heat fluxes via the ray-tracing technique described in Ray Tracing in the Theory Guide. ANSYS Fluent provides several solution parameters in the expanded portion of the **Radiation Model** dialog box (Figure 12.28: The Radiation Model Dialog Box (DTRM) (p. 879)).

Figure 12.28: The Radiation Model Dialog Box (DTRM)



You can control the maximum number of iterations of the radiation calculation during each global iteration by changing the **Maximum Number of Radiation Iterations**. The default setting of 5 means that the radiosity will be updated up to 5 times. The actual number of iterations will be less if the residual convergence criterion is exceeded at any point during these iterations.

The **Residual Convergence Criteria** parameter (0.001 by default) determines when the radiation intensity update is converged. It is defined as the maximum normalized change in the surface intensity from one DTRM radiation iteration to the next (see Equation 12.9 (p. 881)).

You can also control the frequency that the radiation field is updated as the continuous phase solution proceeds. The **Energy Iterations per Radiation Iteration** parameter is set to 10 by default. This means that the radiation calculation is performed once every 10 iterations of the solution process. Increasing the number can speed the calculation process, but may slow overall convergence.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12.3.9.3. S2S Solution Parameters

For the S2S model, as for the DTRM, you can control the frequency with which the radiosity is updated as the continuous-phase solution proceeds. See the previous description of **Energy Iterations per Radiation Iteration**.

If you are using the pressure-based solver and you first solve the flow equations with the energy equation turned off, you should reduce the **Energy Iterations per Radiation Iteration** from 10 to 1 or 2. This will ensure the convergence of the radiosity. If the default value of 10 is kept in this case, it is possible that the flow and energy residuals may converge and the solution will terminate before the radiosity is converged. See Residual Reporting for the S2S Model (p. 881) for more information about residuals for the S2S model.

You can control the maximum number of iterations of the radiation calculation during each global iteration by changing the **Maximum Number of Radiation Iterations**. The default setting of 5 means that the radiosity will be updated up to 5 times. The actual number of iterations will be less if the residual convergence criterion is exceeded at any point during these iterations.

The **Residual Convergence Criteria** (0.001 by default) determines when the radiosity update is converged. It is defined as the maximum normalized change in the radiosity from one S2S radiation iteration to the next (see Equation 12.10 (p. 881)).

12.3.9.4. DO Solution Parameters

For the discrete ordinates model, as for the DTRM, you can control the frequency that the surface intensity is updated as the continuous phase solution proceeds through the **Energy Iterations per Radiation Iteration** setting. See the description of DTRM Solution Parameters (p. 879).

For most problems, the default under-relaxation of 1.0 for the DO equations is adequate. For problems with large optical thicknesses ($aL>10$), you may experience slow convergence or solution oscillation. For such cases, under-relaxing the energy and DO equations is useful. Under-relaxation factors between 0.9 and 1.0 are recommended for both equations.

12.3.9.5. MC Solution Parameters

For the MC model, as for the DTRM, you can control the frequency that the surface intensity is updated as the continuous phase solution proceeds through the **Energy Iterations per Radiation Iteration** setting. See the description of DTRM Solution Parameters (p. 879).

12.3.9.6. Running the Calculation

Once the radiation problem has been set up, you can proceed as usual with the calculation. Note that while the P-1 and DO models will solve additional transport equations and report residuals, the DTRM and the Rosseland and S2S models will not (since they impact the solution only through the energy equation). Residuals for the DTRM and S2S model radiation iterations are reported by ANSYS Fluent every time such an iteration is performed, as described below.

When running the S2S model with the parallel solver, note that coupled walls are encapsulated, which may cause problems with the partitioning of the mesh. See Troubleshooting (p. 2056) for recommendations on correcting such problems.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12.3.9.6.1. Residual Reporting for the P-1 Model

The residual for radiation as calculated by the P-1 model is updated after each iteration and reported with the residuals for all other variables. ANSYS Fluent reports the normalized P-1 radiation residual as defined in Monitoring Residuals (p. 1624) for the other transport equations.

12.3.9.6.2. Residual Reporting for the DO Model

After each DO iteration, the DO model reports a composite normalized residual for all the DO transport equations. The definition of the residuals is similar to that for the other transport equations (see Monitoring Residuals (p. 1624)).

12.3.9.6.3. Residual Reporting for the DTRM

ANSYS Fluent does not include a DTRM residual in its usual residual report that is issued after each iteration. The effect of radiation on the solution can be gathered, instead, via its impact on the energy field and the energy residual. However, each time a DTRM iteration is performed, ANSYS Fluent will print out the normalized radiation error for each DTRM radiation iteration. The normalized radiation error is defined as

$$E = \frac{\sum_{all\ radiating\ surfaces} \left(I_{new} - I_{old} \right)}{N \left(\sigma T^4 / \pi \right)} \tag{12.9}$$

where the error E is the maximum change in the intensity (I) at the current radiation iteration, normalized by the maximum surface emissive power, and N is the total number of radiating surfaces. Note that the default radiation convergence criterion, as noted in DTRM Solution Parameters (p. 879), defines the radiation calculation to be converged when E decreases to 10^{-3} or less.

12.3.9.6.4. Residual Reporting for the S2S Model

ANSYS Fluent does not include an S2S residual in its usual residual report that is issued after each iteration. The effect of radiation on the solution can be gathered, instead, via its impact on the energy field and the energy residual. However, each time an S2S iteration is performed, ANSYS Fluent will print out the normalized radiation error for each S2S radiation iteration. The normalized radiation error is defined as

$$E = \frac{\sum_{all\ radiating\ surface\ clusters} \left(J_{new} - J_{old} \right)}{N \sigma T^4} \tag{12.10}$$

where the error E is the maximum change in the radiosity (J) at the current radiation iteration, normalized by the maximum surface emissive power, and N is the total number of radiating surface clusters. Note that the default radiation convergence criterion, as noted in DTRM Solution Parameters (p. 879), defines the radiation calculation to be converged when E decreases to 10^{-3} or less.

12.3.9.6.5. Disabling the Update of the Radiation Fluxes

Sometimes, you may want to set up your ANSYS Fluent model with the radiation model active and then disable the radiation calculation during the initial calculation phase. For the P-1 and DO models, you can turn off the radiation calculation temporarily by deselecting **P1** or **Discrete Ordinates** in the **Equations** dialog box, which is accessed by right-clicking **Solution Controls** in the tree (under **Solution**) and clicking **Equations...** in the menu that opens. For the DTRM and the S2S model, there is no radiation calculation to disable in the **Equations** dialog box. You can instead set a very large number for **Energy**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Iterations per Radiation Iteration in the **Iteration Parameters** group box of the **Radiation Model** dialog box.

If you turn off the radiation calculation, ANSYS Fluent will skip the update of the radiation field during subsequent iterations, but will leave in place the influence of the current radiation field on energy sources due to absorption, wall heat fluxes, and so on. Turning the radiation calculation off in this way can therefore be used to initiate your modeling work with the radiation model inactive and/or to focus the computational effort on the other equations if the radiation model is relatively well converged.

12.3.10. Postprocessing Radiation Quantities

Information regarding postprocessing radiation quantities can be found in the following sections:

12.3.10.1. Available Variables for Postprocessing
12.3.10.2. Reporting Radiative Heat Transfer Through Boundaries
12.3.10.3. Overall Heat Balances When Using the DTRM
12.3.10.4. Displaying Rays and Clusters for the DTRM
12.3.10.5. Reporting Radiation in the S2S Model

12.3.10.1. Available Variables for Postprocessing

ANSYS Fluent provides radiation quantities that you can use in postprocessing when your model includes the solution of radiative heat transfer. You can generate graphical plots or alphanumeric reports of the following variables/functions:

In the **Radiation...** category:

- **Incident Radiation** (P-1, DO, and MC models)
- **Incident Radiation (Band n)** (non-gray P-1, non-gray DO, non-gray MC models)
- **Absorption Coefficient** (DTRM, P-1, DO, and Rosseland models)
- **Scattering Coefficient** (P-1, DO, and Rosseland models)
- **Refractive Index** (P–1, DO, and Rosseland models)
- **Radiation Temperature** (P-1, DO, and MC models)
- **Surface Cluster ID** (S2S model)
- **Volumetric Absorbed Radiation** (MC model)
- **Volumetric Emitted Radiation** (MC model)
- **Radiation Intensity.Normalized Std Deviation** (MC model)
- **Wall Irradiation Flux.Normalized Std Deviation** (MC model)

In the **Wall Fluxes...** category:

- **Radiation Heat Flux** (all radiation models)
- **Surface Incident Radiation** (S2S, DTRM, and DO models)
- **Absorbed Radiation Flux** (DO model, semi-transparent wall)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Reflected Radiation Flux** (DO model, semi-transparent wall)
- **Transmitted Radiation Flux** (DO model, semi-transparent wall)
- **Beam Irradiation Flux** (DO model, semi-transparent wall)

See Field Function Definitions (p. 1931) for definitions of these postprocessing variables. Note that in addition, incident radiation, transmitted, reflected and absorbed radiation flux are also available on a per-band basis for the non-gray DO model.

Important

- The sign convention on the radiative heat flux is such that the heat flux from the wall surface is a positive quantity.
- It is possible to export heat flux data on wall zones (including radiation) to a generic file that you can examine or use in an external program. See Exporting Heat Flux Data (p. 832) for details.
- Take care not to confuse **Incident Radiation** and **Surface Incident Radiation**. **Incident Radiation** is a volumetric quantity giving the total radiant load passing through the cell (in all directions), whereas **Surface Incident Radiation** is the total radiant load hitting the surface (which will subsequently be absorbed, transmitted and reflected). There is no direct means to report how much radiation has been absorbed/emitted/scattered in cells.

12.3.10.2. Reporting Radiative Heat Transfer Through Boundaries

You can use the **Flux Reports** dialog box to compute the radiative heat transfer through each boundary of the domain, or to sum the radiative heat transfer through all boundaries.

Results → Reports → Fluxes **Edit...**

See Fluxes Through Boundaries (p. 1911) for details about generating flux reports.

12.3.10.3. Overall Heat Balances When Using the DTRM

The DTRM yields a global heat balance and a balance of radiant heat fluxes only in the limit of a sufficient number of rays. In any given calculation, therefore, if the number of rays is insufficient you may find that the radiant fluxes do not obey a strict balance. Such imbalances are the inevitable consequence of the discrete ray tracing procedure and can be minimized by selecting a larger number of rays from each wall boundary.

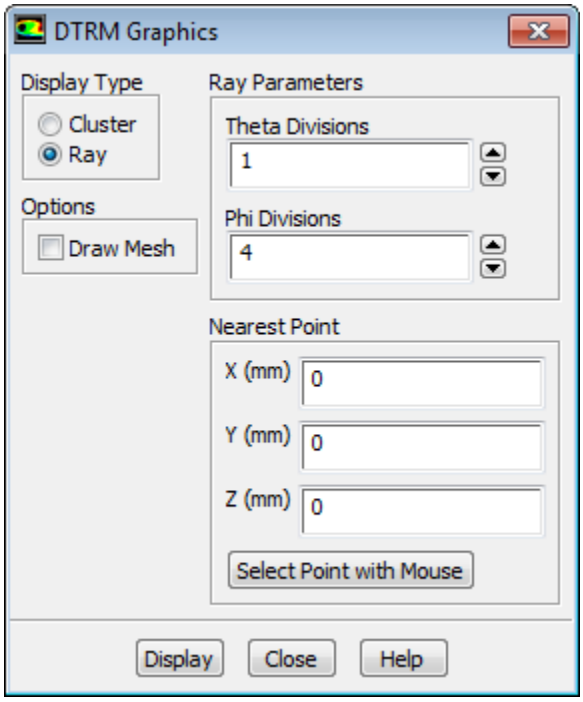
12.3.10.4. Displaying Rays and Clusters for the DTRM

When you use the DTRM, ANSYS Fluent allows you to display surface or volume clusters, as well as the rays that emanate from a particular surface cluster. You can use the **DTRM Graphics** dialog box (Figure 12.29: The DTRM Graphics Dialog Box (p. 884)) for all of these displays.

Postprocessing → Model Specific → DTRM Graphics...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 12.29: The DTRM Graphics Dialog Box



12.3.10.4.1. Displaying Clusters

To view clusters, select **Cluster** under **Display Type** and then select either **Surface** or **Volume** under **Cluster Type**.

To display all of the surface or volume clusters, select the **Display All Clusters** option under **Cluster Selection** and click the **Display** button.

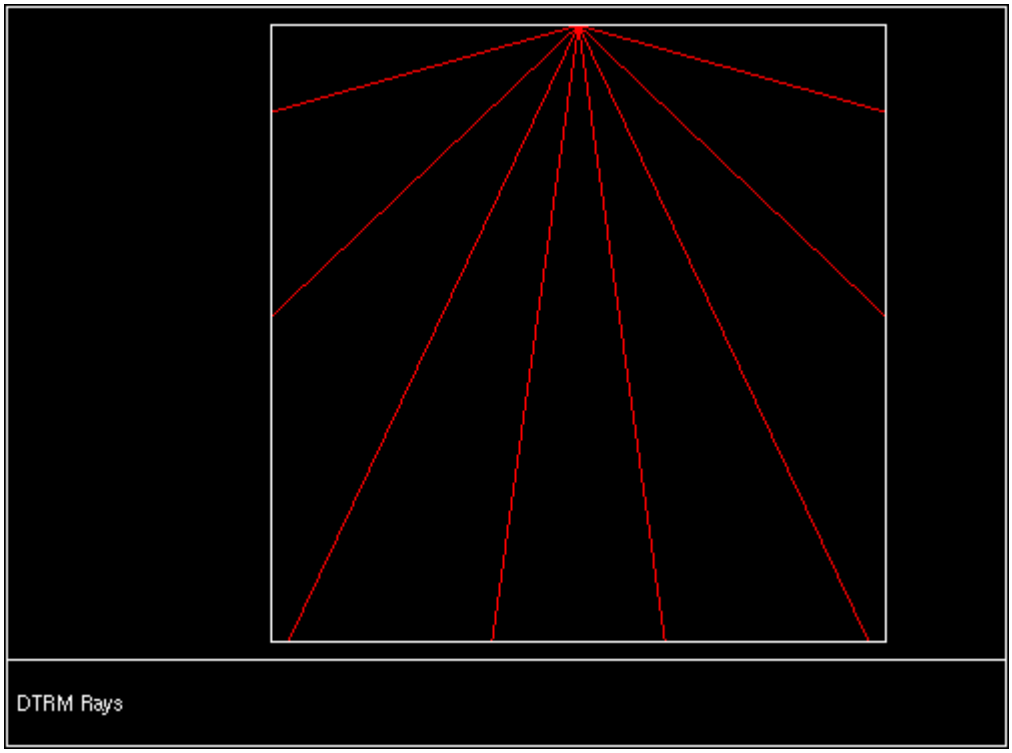
To display only the cluster (surface or volume) nearest to a specified point, deselect the **Display All Clusters** option and specify the coordinates under **Nearest Point**. You may also use the mouse to choose the nearest point. Click the **Select Point With Mouse** button and then right-click a point in the graphics window.

12.3.10.4.2. Displaying Rays

To display the rays emanating from the surface cluster nearest to the specified point, select **Ray** under **Display Type**. Set the appropriate values for **Theta** and **Phi Divisions** under **Ray Parameters** (see Setting Up the DTRM (p. 848) for details), and then click the **Display** button. Figure 12.30: Ray Display (p. 885) shows a ray plot for a simple 2D geometry.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 12.30: Ray Display



12.3.10.4.3. Including the Mesh in the Display

For some problems, especially complex 3D geometries, you may want to include portions of the mesh in your ray or cluster display as spatial reference points. For example, you may want to show the location of an inlet and an outlet along with displaying the rays. This is accomplished by enabling the **Draw Mesh** option in the **DTRM Graphics** dialog box. The **Mesh Display** dialog box will appear automatically when you enable the **Draw Mesh** option, and you can set the mesh display parameters there. When you click **Display** in the **DTRM Graphics** dialog box, the mesh display, as defined in the **Mesh Display** dialog box, will be included in the ray or cluster display.

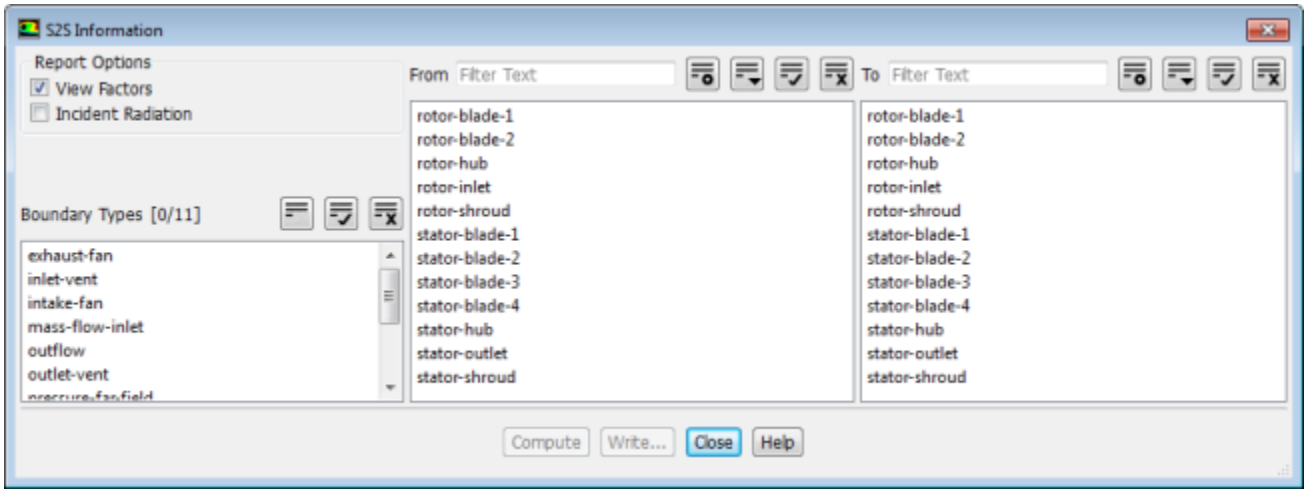
12.3.10.5. Reporting Radiation in the S2S Model

When you use the S2S model, ANSYS Fluent allows you to view the values of the view factor and radiation emitted from one zone to any other zone. You can use the **S2S Information** dialog box (Figure 12.31: The S2S Information Dialog Box (p. 886)) to generate a report of these values in the console or as a separate file.

Postprocessing → Model Specific → S2S Information...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 12.31: The S2S Information Dialog Box



The steps for generating the report are as follows:

1. Specify the values in which you are interested by selecting **View Factors** and/or **Incident Radiation**.

2. Choose the zones for which you would like data by selecting them in the lists under **From** and **To** (at least one zone must be selected under each list). To select all of the zones of a particular type, click that category in the list under **Boundary Types**.

3. Specify how you would like to present the data. To report the values in the console, click the **Compute** button. To write the data as an S2S Info File (`.sif` format), click the **Write...** button and enter a file name in The Select File Dialog Box (p. 19).

The following is an example of how the data is presented:

```
S2S Information
From wall1 to:
                    View factor       Incident Radiation
          wall1          0.0000                  0.0000
          wall2          0.2929             171387.7813
          wall3          0.2929             155305.7969
          wall4          0.4142              29055.9023

From wall2 to:
                    View factor       Incident Radiation
          wall1          0.2929             306451.9688
          wall2          0.0000                  0.0000
          wall3          0.4142             214195.0938
          wall4          0.2929              19153.2715
```

Note that the header listed above (`S2S Information`) is not displayed in the console.

12.3.11. Solar Load Model

ANSYS Fluent provides a solar load model that can be used to calculate radiation effects from the sun's rays that enter a computational domain. Two options are available for the model: solar ray tracing and DO irradiation. The ray tracing approach is a highly efficient and practical means of applying solar loads as heat sources in the energy equations. In cases where you want to use the discrete ordinates (DO) model to calculate radiation effects within the domain, an option is available to supply outside beam direction and intensity parameters directly to the DO model. The solar load model includes a solar calculator utility that can be used to construct the sun's location in the sky for a given time-of-day, date,

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and position. Solar load is available in the 3D solver only, and can be used to model steady and unsteady flows.

12.3.11.1. Introduction

Typical applications that are well-suited for solar load simulations include the following:

- automotive climate control (ACC) applications
- human comfort modeling applications in buildings

The effects of solar loading are needed in many ACC applications, where the temperature, humidity, and velocity fields around passengers (and drivers) are desired. ACC systems are tested for their capacity to cool down passenger compartments after they have been "soaked" in intense solar radiation. ANSYS Fluent's solar load model will enable you to simulate solar loading effects and predict the time it will take to reasonably cool down the cabin of a car that has been exposed to solar radiation, as well as predict the time interval needed to lower the temperature in specified points and areas within the domain.

In the analysis of buildings, solar loading provides a significant burden on the cooling requirement in warm climates, particularly where architects want to use the aesthetics of glazed facades. Even in cooler climates, solar loading can provide a burden during warmer seasons where modern buildings are well insulated against thermal loss during winter months. As well as providing an engineer with a practical tool for determining the solar heating effect inside a building, ANSYS Fluent's solar load model will allow the solar transmission through all glazed surfaces to be determined over the course of a day, allowing important decisions to be made before undertaking any flow studies. ANSYS Fluent's solar load model also allows you to simulate porous blinds, which can transmit a portion of the solar radiation while also allowing fluid flow.

12.3.11.2. Solar Ray Tracing

The solar load model's ray tracing algorithm can be used to predict the direct illumination energy source that results from incident solar radiation. It takes a beam that is modeled using the sun position vector and illumination parameters, applies it to any or all wall, porous jump, and inlet/outlet boundary zones that you specify, performs a face-by-face shading analysis to determine well-defined shadows on all boundary faces and interior walls, and computes the heat flux on the boundary faces that results from the incident radiation.

Important

The solar ray tracing model includes only boundary zones that are adjacent to fluid zones in the ray tracing calculation. Therefore, boundary zones that are attached to solid zones are treated as transparent and are not included in the ray tracing algorithm.

The resulting heat flux that is computed by the solar ray tracing algorithm is coupled to the ANSYS Fluent calculation via a source term in the energy equation. The heat sources are added directly to computational cells bordering each face and are assigned to adjacent cells in the following order: shell conduction cells, solid cells, and fluid cells. Heat sources are assigned to one of these types of adjacent cells, only. You can choose to override this order and include adjacent fluid cells in the solar load calculation by issuing a command in the text user interface (see Text Interface-Only Commands (p. 907) for details). Note that the sun position vector and solar intensity can be entered either directly by you or computed from the solar calculator. Direct and diffuse irradiation parameters can also be specified using a user-defined function (UDF) and hooked to ANSYS Fluent in the **Radiation Model** dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The solar ray tracing option allows you to include the effects of direct solar illumination as well as diffuse solar radiation in your ANSYS Fluent model. A two-band spectral model is used for direct solar illumination and accounts for separate material properties in the visible and infrared bands. A single-band hemispherical-averaged spectral model is used for diffuse radiation. Opaque materials are characterized in terms of two-band absorptivities. A semi-transparent material requires specification of absorptivity and transmissivity. Values that you specify for transmissivity and absorptivity are defined for normal incident rays. ANSYS Fluent recomputes/interpolates these values for the given angle of incidence.

The solar ray tracing algorithm also accounts for internal scattered and diffusive loading. The reflected component of direct solar irradiation is tracked. A fraction of this radiative heat flux, called internally scattered energy is applied to all the surfaces participating in the solar load calculation, weighted by area. The internally scattered energy depends on the scattering fraction that is specified in the TUI, whose default value is 1. Depending on the reflectivity of the primary surface, the scattering fraction can be responsible for the inclusion (or exclusion) of a large amount of radiation within the rest of the domain.

Also included as internally scattered energy is the contribution of the transmitted component of diffuse solar irradiation (which enters a domain through semi-transparent walls depending upon the hemispherical transmissivity). The total value of internally scattered energy is reported to the ANSYS Fluent console. The ambient flux is obtained by dividing the internally scattered energy by the total surface area of the faces participating in the solar load calculation.

Note that **Solar Ray Tracing** is *not* a participating radiation model. It does not deal with emission from surfaces, and the reflecting component of the primary incident load is distributed uniformly across all surfaces rather than being local to the surfaces reflected to. If surface emission is an important factor in your case then you can consider implementing a radiation model (for example, P-1) in conjunction with **Solar Ray Tracing**.

Note the following limitation when using the solar ray tracing model:

- Non-conformal interfaces are not supported with the solar ray tracing model.

12.3.11.2.1. Shading Algorithm

The shading calculation that is used for solar ray tracing is a straightforward application of vector geometry. A ray is traced from the centroid of a test face in the direction of the sun. Every other face is checked to determine if the ray intersects the candidate face and if the candidate face is in front of the test face. If both conditions are met, then an opaque face completely shades the test face. A semi-transparent face attenuates the incident energy.

A Barycentric coordinate formulation is used to construct triangle-ray intersections. A quadrilateral ray intersection method is used to handle the case when model surfaces contain quadrilaterals. A quad-tree preprocessing step is applied to reduce the ray tracing algorithm complexity that can lead to long run times for 10^4 faces and greater. The quad-tree refinement factor can be modified in the text interface. The default value of this parameter is 7, which is sufficient to cover the entire spectrum of mesh sizes between one cell and five million cells. If the mesh is greater than five million cells, an increase in this parameter would reduce the CPU time needed to compute the solar loads.

12.3.11.2.2. Glazing Materials

Incident solar radiation can be applied to glass and plastic glazing materials of various types at wall boundaries, and the effects of coated glazings modeled using the solar ray tracing algorithm. To model solar optical properties, you will need to specify the transmissivity and reflectivity of the material in the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Wall boundary conditions dialog box. You can obtain these values from the glass (or plastic) manufacturer or use data from another source (for example, ASHRAE Handbook).

Glazing optical properties are dependent on incident angle, and the variation is significant for an incident angle greater than 40 degrees. As the incident angle increases from zero, transmissivity decreases, reflectivity increases, and absorptivity increases initially due to lengthened optical path, and then decreases as more incident radiation is reflected. The shape of the property curve varies with glass type and thickness. This difference is more pronounced for coated glass or for a multiple-pane glazing system. It cannot be assumed that all glazing systems have a universal angular dependence.

For coated glazings, the spectral transmissivity and reflectivity at any incident angle are approximated in the solar load model from the normal angle of incidence [31] (p. 2888).

Transmissivity is given by

$$T(\theta,\lambda)=T(0,\lambda)Tref(\theta) \tag{12.11}$$

where

$$Tref(\theta)=a0+a1cos(\theta)+a^2cos(\theta^2)+a3cos(\theta^3)+a4cos(\theta^4) \tag{12.12}$$

Reflectivity is given by

$$R(\theta,\lambda)=R(0,\lambda)[1-Rref(\theta)]+Rref(\theta) \tag{12.13}$$

where

$$Rref(\theta)=b0+b1cos(\theta)+b^2cos(\theta^2)+b3cos(\theta^3)+b4cos(\theta^4)-Tref(\theta) \tag{12.14}$$

The constants used in Equation 12.11 (p. 889) and Equation 12.13 (p. 889) are for coated glazings and are taken from Finlayson and Arasteh. [31] (p. 2888). The normal transmissivity and reflectivity, $T(0,\lambda)$ and $R(0,\lambda)$ are specified in the **Wall** boundary conditions dialog box.

12.3.11.2.3. Inputs

The following inputs are required for the solar ray tracing algorithm:

- sun direction vector
- direct solar irradiation
- diffuse solar irradiation
- spectral fraction
- direct and IR absorptivity (opaque wall)
- direct and IR absorptivity and transmissivity (semi-transparent wall and porous jump)
- diffuse hemispherical absorptivity and transmissivity (semi-transparent wall)
- solar transmissivity factor
- quad tree refinement factor
- scattering fraction

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- ground reflectivity

The sun direction vector is the direction vector looking to the sun, from which the direct irradiation will be incident. You can enter the vector components (**X**, **Y**, **Z**) and the direct and diffuse solar irradiation fluxes in the **Radiation Model** dialog box, or you can have these parameters derived from the solar calculator. These irradiation fluxes can also be specified using a user-defined function (User-Defined Functions (UDFs) for Solar Load (p. 894)). The spectral fraction is the final input in the **Radiation Model** dialog box. This defines the split of visible and infra-red (shortwave and longwave respectively) radiation, specifically the fraction of the direct irradiation flux that is in the visible band. These quantities can also be defined through the text interface.

The scattering fraction defines the amount of non-absorbed radiation that will be distributed (uniformly) across all participating surfaces. This is required as the solar load model does not track the rays beyond the first opaque surface. Therefore, a highly glazed space where incident radiation is likely to be reflected back out will have a low value. Conversely, a predominantly opaque (wall-bounded) space where reflected radiation is likely to be incident upon (and ultimately absorbed by) other opaque surfaces will have a high value. This parameter is defined through the text interface only, taking a default value of 1.0:

`define` → `models` → `radiation` → `solar-parameters` → `scattering-fraction`

The ground reflectivity is used by the solar calculator to compute the background diffuse radiation intensity component contributed to by radiation reflected off the ground. This should be based on typical figures for the surface reflectivity of the outside ground surfaces. By default this is set to 0.2, but can be adjusted through the text-interface:

`define` → `models` → `radiation` → `solar-parameters` → `ground-reflectivity`

The quad-tree-refinement parameter determines the level of detail used by the shading algorithm. By default this is set to 7 which will generally work well, but can lie between 0 and 10. This is defined only through the text interface:

`define` → `models` → `radiation` → `solar-parameters` → `quad-tree-refinement`

Further details on the text interface-only entries is provided later in this section (see Text Interface-Only Commands).

The absorptivity and transmissivity parameters related to a wall or porous jump are entered in the **Wall** (under the **Radiation** tab) or **Porous Jump** boundary condition dialog box, respectively, for the particular boundary zones you want to participate in solar ray tracing. On flow boundaries you have a solar transmissivity factor to allow you to attenuate the incoming solar flux, for example, set to 1 for a fully open inlet or set to 0 for a light obscuring louvered inlet.

12.3.11.3. DO Irradiation

The solar load model's discrete ordinates (DO) irradiation option provides you with an easy means of applying a solar load directly to the DO model. Unlike the ray tracing solar load option, the DO irradiation method does not compute heat fluxes and apply them as heat sources to the energy equation. Instead, the irradiation flux is applied directly to semi-transparent walls (which you specify) as a boundary condition, and the radiative heat transfer is derived from the solution of the DO radiative transfer equation.

The following inputs are required for DO irradiation at semi-transparent walls:

- direct irradiation

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- diffuse irradiation
- beam direction
- beam width
- diffuse fraction

In the **Wall** boundary condition dialog box for each semi-transparent wall you want to participate in DO irradiation, you can specify that the beam direction, direct irradiation, and diffuse irradiation be derived from the solar parameters (for example, solar calculator) that you set (or compute) in the **Radiation Model** dialog box. This is done by checking the **Use Beam Direction from Solar Load Model Settings** and **Use Direct and Diffuse Irradiation from Solar Load Model Settings** boxes. When selected, ANSYS Fluent sets the beam width (the angle subtended by the sun) to the default value of 0.53 degrees for DO irradiation.

Important

Note that the sign of the beam direction that is needed for the DO model is opposite the sun direction vector that is entered or derived from the solar parameters. The beam direction in the DO model is the direction of external radiation (for example, radiation coming from the sun), while the sun direction vector in the solar load model points to the sun. Incident radiation and the sun angle always have an opposite sign since they are quantities that are defined from opposite perspectives.

12.3.11.4. Solar Calculator

ANSYS Fluent provides a solar calculator that can be used to compute solar beam direction and irradiation for a given time, date, and position. These values can be used as inputs to the solar ray tracing algorithm or as semi-transparent wall boundary conditions for discrete ordinates (DO) irradiation.

12.3.11.4.1. Inputs/Outputs

Inputs needed for the solar calculator are:

- global position (latitude, longitude, time zone)
- starting date and time
- mesh orientation
- solar irradiation method
- sunshine factor

Global position consists of latitude, longitude, and time zone (relative to GMT). The time of day for a transient simulation is the starting time plus the flow-time. For mesh orientation, you will need to specify the North and East direction vector in the CFD mesh. The default solar irradiation method is Fair Weather Conditions. Alternatively, you can choose the Theoretical Maximum method. The sunshine factor is simply a linear reduction factor for the computed incident load that allows for cloud cover to be accounted for, if appropriate.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can specify these inputs in the **Solar Calculator** dialog box that is accessible from the **Radiation Model** dialog box (Figure 12.35: The Solar Calculator Dialog Box (p. 898)). Alternatively, you can enter the parameters using text interface commands (Additional Text Interface Commands (p. 909)).

The following values are computed by the solar calculator and are displayed in the console whenever the solar calculator is used:

- sun direction vector
- sunshine fraction
- direct normal solar irradiation at earth's surface
- diffuse solar irradiation - vertical and horizontal surface
- ground reflected (diffuse) solar irradiation - vertical surface

Direct normal solar irradiation is computed using the ASHRAE Fair Weather Conditions method, when this option is selected in the solar calculator. Note: Equation 20 and Table 7 from Chapter 30 of the 2001 ASHRAE Handbook of Fundamentals. The theoretical maximum values for direct normal solar irradiation and diffuse solar irradiation are computed using NREL's Theoretical Maximum method, when this option is selected. In practice, these values are unlikely to be experienced due to atmospheric conditions.

ANSYS Fluent computes the diffuse solar irradiation components (vertical and horizontal) internally for each face in the domain. When the Theoretical Maximum method is chosen, these diffuse irradiation values provide estimates for the maximum vertical and horizontal surface effects.

12.3.11.4.2. Theory

ANSYS Fluent provides two options for computing the solar load: Fair Weather Conditions method and Theoretical Maximum method. Although these methods are similar, there is a key difference. The Fair Weather Conditions method imposes greater attenuation on the solar load, which is representative of atmospheric conditions that are fair–but not completely clear.

The equation for normal direct irradiation applying the Fair Weather Conditions Method is taken from the ASHRAE Handbook:

$$Edn=\frac{A}{e^{\frac{B}{\sin(\beta)}}} \tag{12.15}$$

where A and B are apparent solar irradiation at air mass $m=0$ and atmospheric extinction coefficient, respectively. These values are based on the earth's surface on a clear day. β is the solar altitude (in degrees) above the horizontal.

The equation for direct normal irradiation that is used for the Theoretical Maximum Method is taken from NREL's Solar Position and Intensity Code (Solpos):

$$Edn=S_{etrn}S_{unprime} \tag{12.16}$$

where S_{etrn} is the top of the atmosphere direct normal solar irradiance and $S_{unprime}$ is the correction factor used to account for reduction in solar load through the atmosphere.

The calculation for the diffuse load in the solar model is based on the approach suggested in the 2001 ASHRAE Fundamental Handbook (Chapter 20, Fenestration). The equation for diffuse solar irradiation on a vertical surface is given by:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$Ed = CYEdn \tag{12.17}$$

where C is a constant whose values are given in Table 7 from Chapter 30 of the 2001 ASHRAE Handbook of Fundamentals, Y is the ratio of sky diffuse radiation on a vertical surface to that on a horizontal surface (calculated as a function of incident angle), and Edn is the direct normal irradiation at the earth's surface on a clear day.

The equation for diffuse solar irradiation for surfaces other than vertical surfaces is given by:

$$Ed = CEdn\frac{\left(1+\cos\varepsilon\right)}{2} \tag{12.18}$$

where ε is the tilt angle of the surface (in degrees) from the horizontal plane.

The equation for ground reflected solar irradiation on a surface is given by:

$$Er = Edn\left(C+\sin\beta\right)\rho_g\frac{\left(1-\cos\varepsilon\right)}{2} \tag{12.19}$$

where ρ_g is the ground reflectivity. The total diffuse irradiation on a given surface will be the sum of Ed and Er when the input for diffuse solar radiation is taken from the solar calculator. Otherwise, if the `constant` option is selected in the **Radiation** dialog box, then the total diffuse irradiation will be the same as specified in the dialog box.

12.3.11.4.3. Computation of Load Distribution

In calculating the solar load that will be incident on each surface, it is necessary to distinguish between the calculation of diffuse and direct solar loads. A direct load will be tracked from participating transmissive boundary surfaces and non-participating boundary surfaces, the former provides some opportunity to attenuate the incoming flux by absorption and reflection, while the non-participating surfaces allow the flux to enter without any drop in intensity. The direct load is then tracked through the model space until it is incident on an opaque surface, or it exists through a transmissive or non-participating boundary zone. During its passage, its intensity will be attenuated as it passes through participating semi-transparent internal walls, where some radiation may be absorbed and some may be reflected. The total amount of direct radiation that is reflected at internally facing surfaces will be added to the scattered radiation budget for further use later.

The diffuse load originates at participating transmissive boundary surfaces. It is these surfaces that permit diffuse radiation to enter, irrespective of their orientation relative to the direction vector. For each transmissive surface, some of the incoming diffuse load may be immediately absorbed and/or reflected to the outside. The rest is assumed to be transmitted inside and summed from all of these surfaces to give an initial diffuse budget. Onto this budget is added a fraction of the previously computed scattered radiation from the direct load, the fraction used is defined as an input to the model. This provides the total diffuse load. This is then uniformly distributed across all surfaces that are participating in the solar calculation, irrespective of whether they are opaque or semi-transparent. There is no scope to define local absorptivity for this distribution and no biasing with regards proximity to transmissive surfaces. Note that a non-participating boundary zone will allow direct load to enter the model space but will not provide an incoming quantity of diffuse load.

Note that the solar flux that is externally incident on an opaque surface will be completely disregarded, for example solar load on an opaque roof of a model whose internals only are modeled will not be included as a heat gain. Instead, this heat gain should be manually calculated and applied as a thermal condition, typically using a fixed heat flux or a radiation/mixed condition.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12.3.11.5. Using the Solar Load Model

When you want to run a steady-state solution with solar load enabled, you simply set up the solar load model (Setting Up the Solar Load Model (p. 894)) and boundary conditions (Setting Boundary Conditions for Solar Loading (p. 899)) for your case, and then run the simulation. The solution data file will contain the solar fluxes that you can use for postprocessing. For a steady-state solution, the solar loads are computed on initialization. If you want to initially solve a case without solar loading (say, for stability) and then add the effects of solar loading afterward, you will need to enable the solar load model through the text user interface (TUI).

Important

Note that you can compute the solar load at any time once you have set up the model by using the `sol-on-demand` text interface command (see Additional Text Interface Commands (p. 909) for details).

When you want to run a transient solar load simulation, the process is the same as for the steady-state case but you will need to specify the additional **Time Steps per Solar Load Update** parameter in the **Radiation Model** dialog box. ANSYS Fluent will re-compute the sun position and irradiation and update solar loads with this specified frequency.

Important

Note that parallel simulations involving the solar load model are set up and computed using the same steps as in serial.

12.3.11.5.1. User-Defined Functions (UDFs) for Solar Load

You can write a user-defined function (UDF) to specify direct and diffuse solar intensity using the `DEFINE_SOLAR_INTENSITY` macro. See `DEFINE_SOLAR_INTENSITY` in the for more information. After it is interpreted or compiled, you can hook your intensity UDF for direct or diffuse solar irradiation by selecting `user-defined` in the drop-down lists for these parameters in the **Radiation Model** dialog box. See Step 2 in Setting Up the Solar Load Model (p. 894) for details.

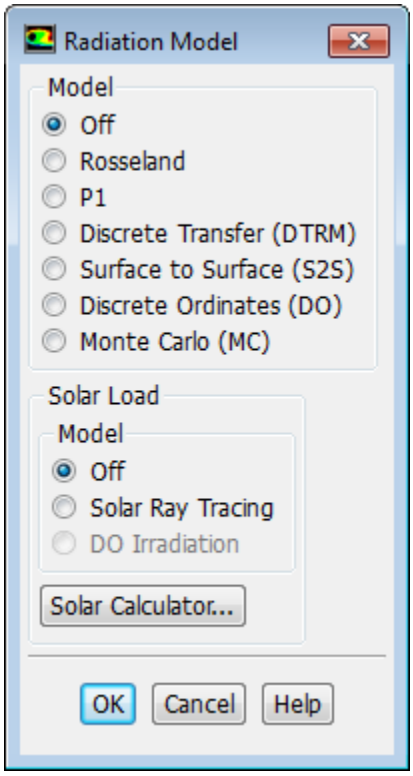
12.3.11.5.2. Setting Up the Solar Load Model

The solar load model is enabled in the **Radiation Model** dialog box (Figure 12.32: The Radiation Model Dialog Box (p. 895)).

Setup → Models → Radiation → Edit...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 12.32: The Radiation Model Dialog Box



Important

Solar load is available in the 3D solver only, and can be used to model steady and unsteady flows.

The solar load model has two options: **Solar Ray Tracing** and **DO Irradiation**. **Solar Ray Tracing** can be applied as a stand-alone solar loading model, or it can be used in conjunction with one of the ANSYS Fluent radiation models (P1, Rosseland, Discrete Transfer, Surface-to-Surface, Discrete Ordinates). **DO Irradiation** is available only when the **Discrete Ordinates (DO)** radiation model is enabled.

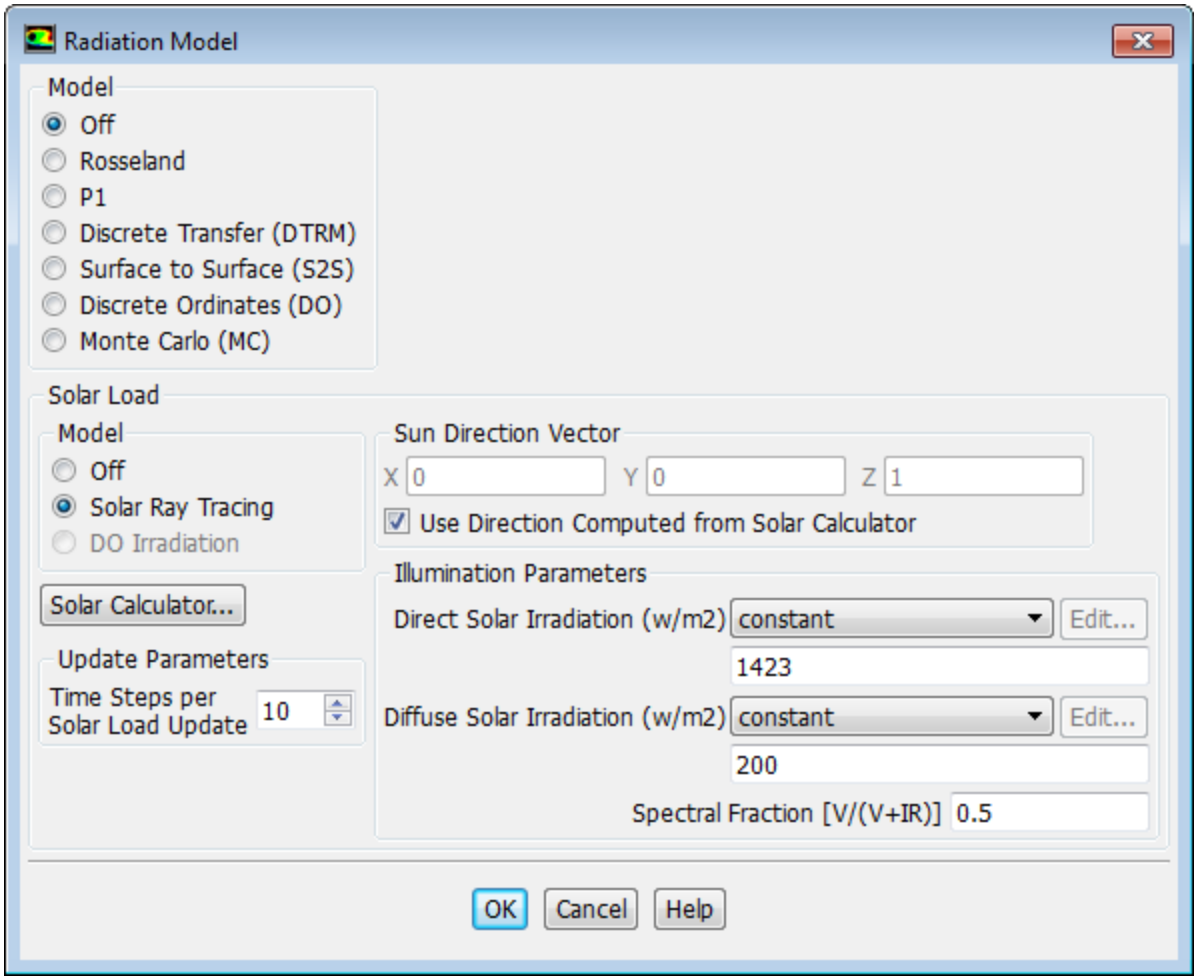
To set up the solar load model, perform the following steps:

1. Enable the solar load model in the **Radiation Model** Dialog Box.

   a. To enable the solar ray tracing algorithm, select **Solar Ray Tracing** under **Solar Load** (Figure 12.33: The Radiation Model Dialog Box (With Solar Load Model Solar Ray Tracing Option) (p. 896)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

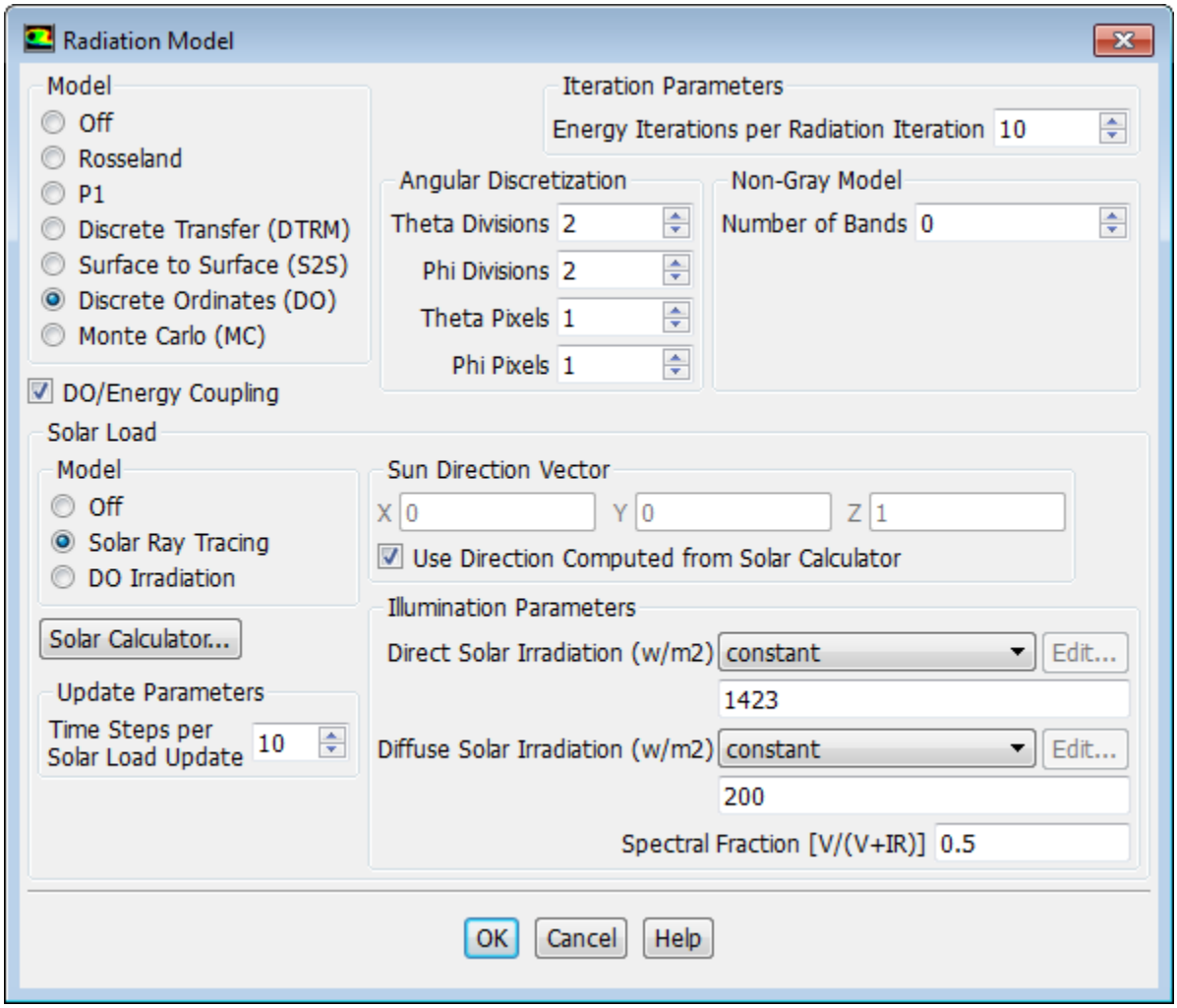
Figure 12.33: The Radiation Model Dialog Box (With Solar Load Model Solar Ray Tracing Option)



b. To enable the DO irradiation option, first select **Discrete Ordinates** under **Model**, and then select **DO Irradiation** under **Solar Load** (Figure 12.34: The Radiation Model Dialog Box (with Solar Load Model DO Irradiation Option) (p. 897)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 12.34: The Radiation Model Dialog Box (with Solar Load Model DO Irradiation Option)



2. Define the solar parameters.

   a. Enter values for the **X**, **Y**, and **Z** components of the **Sun Direction Vector**. Alternatively, you can choose to have this vector computed from the solar calculator by enabling the **Use Direction Computed from Solar Calculator** option.

   b. Specify the illumination parameters.

      i. Enter a value for **Direct Solar Irradiation** under **Illumination Parameters**. This parameter is the amount of energy per unit area in W / m^2 due to direct solar irradiation. This value may depend on the time of year and the clearness of the sky. Make your selection in the drop-down list next to **Direct Solar Irradiation** and either enter a **constant** value, have the value computed from the **solar calculator**, or specify it using a **user-defined function**. (For more information on writing solar intensity UDFs, see `DEFINE_SOLAR_INTENSITY` in the Fluent Customization Manual.) For transient simulations, you have the additional option of specifying a time-dependent **piecewise-linear** and **polynomial** profile for direct solar irradiation.

      ii. Enter a value for **Diffuse Solar Irradiation**, which is the amount of energy per unit area in W / m^2 due to diffuse solar irradiation. This value may depend on the time of year, the clearness of the sky, and also on ground reflectivity. Make your selection in the drop-down list next to **Diffuse Solar Irradiation** and either enter a **constant** value, have the value computed from the **solar calculator**,

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

or specify it using a **user-defined function**. (For more information on writing solar intensity UDFs, see DEFINE_SOLAR_INTENSITY in the .) For transient simulations, you have the additional option of specifying a time-dependent **piecewise-linear** and **polynomial** profile for diffuse solar irradiation.

iii. If you are using the **Solar Ray Tracing** solar load model (Figure 12.33: The Radiation Model Dialog Box (With Solar Load Model Solar Ray Tracing Option) (p. 896)), then you will need to enter a value for **Spectral Fraction**. The spectral fraction is the fraction of incident solar radiation in the visible part of the solar radiation spectrum. The spectral fraction is not used for DO irradiation since the DO implementation is intended only for a single band.

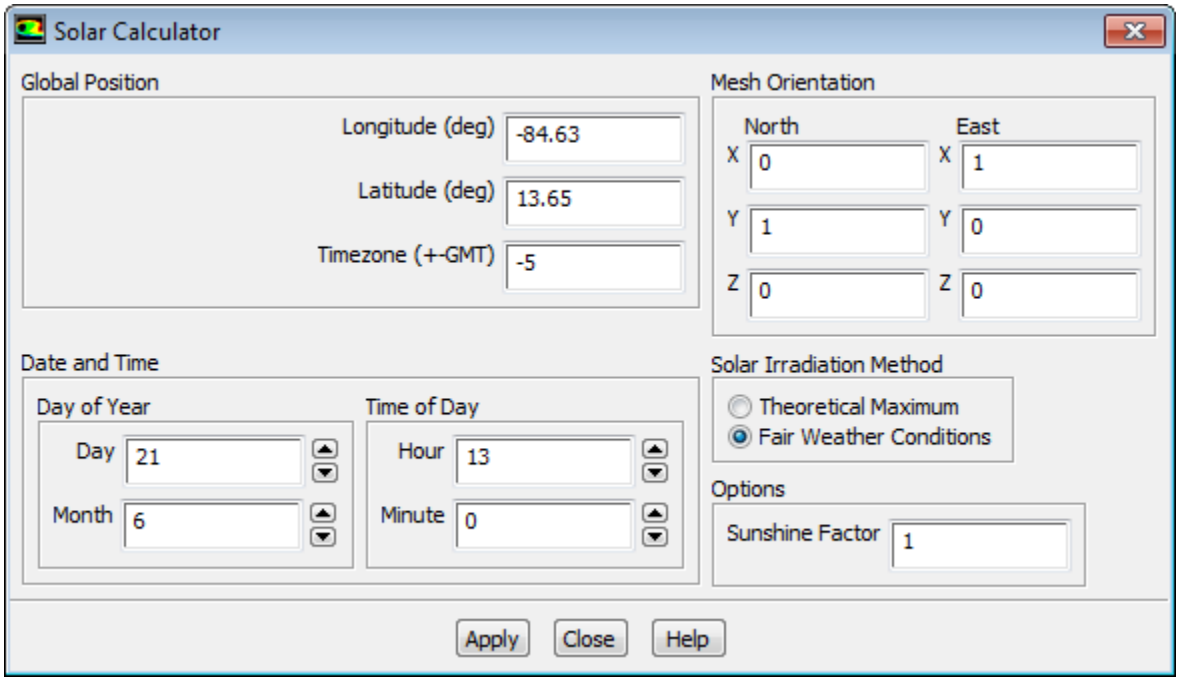
$$\text{Spectral Fraction} = \frac{V}{V+IR} \tag{12.20}$$

where V is the visible incident solar radiation, and $V+IR$ is the total incident solar radiation (visible plus infrared).

3. Use the solar calculator to compute solar beam direction and irradiation.

a. Click **Solar Calculator...** in the **Radiation Model** dialog box to open the **Solar Calculator** dialog box (Figure 12.35: The Solar Calculator Dialog Box (p. 898)).

Figure 12.35: The Solar Calculator Dialog Box



b. In the **Solar Calculator** dialog box, define the **Global Position** by the following parameters:

i. Enter a real number in degrees for **Longitude**. Values may range from -180 to 180 where negative values indicate the Western hemisphere and positive values indicate the Eastern hemisphere.

ii. Enter a real number for **Latitude** in degrees. Values can range from -90° (the South Pole) to 90° (the North Pole), with 0° defined as the equator.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

iii. Enter an integer for **Timezone** that is the local time zone in hours relative to Greenwich Mean Time (+-GMT). This value can range from $+12$ to -12.

Important

Note that you must specify all three **Global Position** parameters for the solar calculator.

---

c. Define the local **Date and Time** by the following parameters:

i. Enter an integer for **Day** and **Month** under **Day of Year**.

ii. Enter an integer for **Hour** that ranges from 0 to 24 under **Time of Day**. Enter an integer or floating point number for **Minute**.

The time of day is based on a 24-hour clock: 0 hours and 0 minutes corresponds to 12:00 a.m. and 23 hours 59.99 min corresponds to 11:59.99 p.m. For example, if the local time was 12:01:30 a.m., you would enter `0` for **Hour** and `1.5` for **Minute**. If the local time was 4:17 p.m., you would enter `16` for **Hour** and `17` for **Minute**.

d. Define the **Mesh Orientation** as the vectors for North and East in the CFD mesh system of coordinates.

e. Select the appropriate **Solar Irradiation Method**. The **Fair Weather Conditions** is the default method.

f. Enter an integer for **Sunshine Fraction** (default = 1).

g. Click **Apply**.

The solar calculator output parameters are computed and the results are reported in the console. The default values are shown below:

```
Fair Weather Conditions:
 Sun Direction Vector: X: -0.0785396, Y: 0.170758, X: 0.982178
 Sunshine Fraction: 1
 Direct Normal Solar Irradiation (at Earth's surface) [W/m^2]:
 881.635
 Diffuse Solar Irradiation - vertical surface: [W/m^2]:
 152.107
 Diffuse Solar Irradiation - horizontal surface: [W/m^2]:
 118.727
 Ground Reflected Solar Irradiation - vertical surface: [W/m^2]:
 96.4649
```

4. For transient simulations, enter the **Time Steps Per Solar Load Update** under **Update Parameters**. The number of time steps that you specify will direct the ANSYS Fluent solver to update the solar load data for the specified flow-time intervals in the unsteady solution process.

12.3.11.5.3. Setting Boundary Conditions for Solar Loading

Once you have defined the solar parameters for the solar load model (Setting Up the Solar Load Model (p. 894)), you will need to set up boundary conditions for boundary zones that will participate in solar loading. The boundary condition dialog boxes can be opened by right-clicking the boundary name in the tree (under **Setup/Boundary Conditions**) and clicking **Edit...** in the menu that opens; alternatively, you can open them from the **Boundary Conditions** task page:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → Boundary Conditions

12.3.11.5.4. Solar Ray Tracing

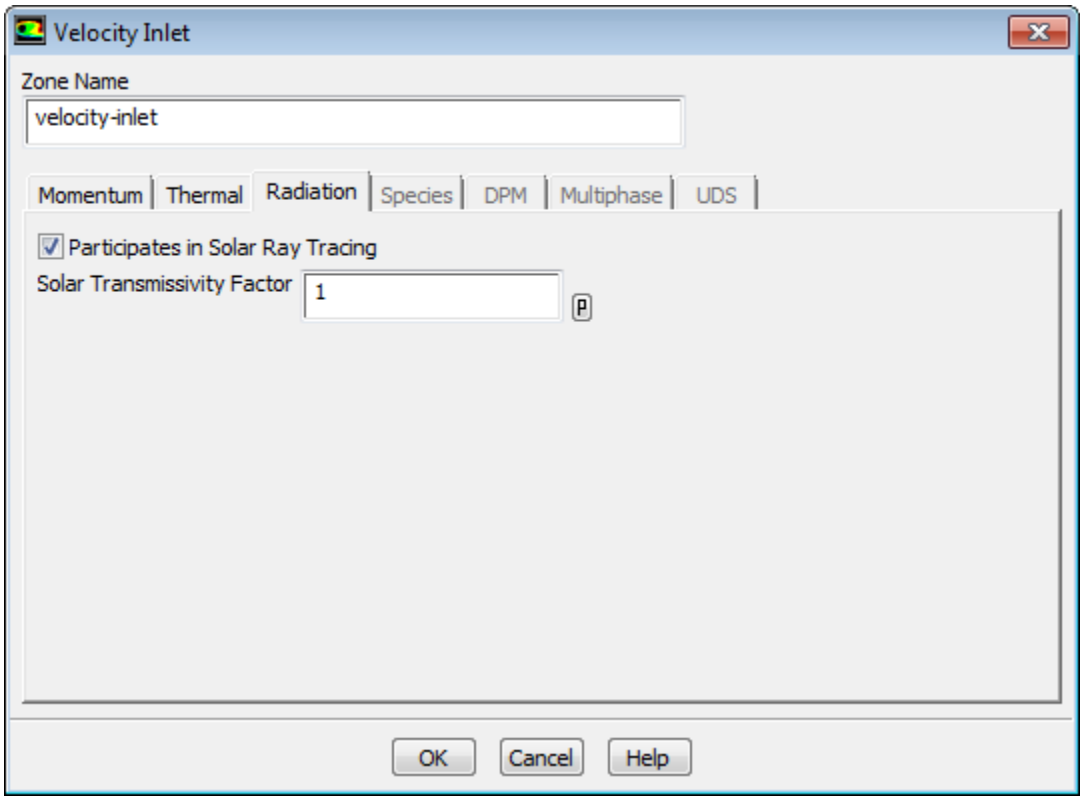
Note

Solar ray tracing is designed to be applied to your geometrical information, therefore, you must include every side of the external, or outer, enclosure/boundary for ray-tracing purposes, unless you expect that solar irradiation cannot enter through such boundaries. If you exclude any internal zone from the solar load computation, then this would mean that the resulting obstruction or blockage has been removed which would also be equivalent to a transparent surface. Excluded walls will also not participate in the computation of scattered energy by the solar load model and, therefore, do not gain any heat from scattered energy.

1. Set the boundary condition for each inlet and exit boundary zone that you want to include in solar loading.

   a. Open the inlet or exit boundary condition dialog box (for example, **Velocity Inlet**) and click the **Radiation** tab (Figure 12.36: The Velocity Inlet Dialog Box (p. 900)).

   Figure 12.36: The Velocity Inlet Dialog Box



   b. Enable the **Participates in Solar Ray Tracing** option (this option is enabled for all boundary conditions by default). If you deactivate solar ray tracing by disabling this option the surface will be ignored and the solar ray will pass through it with no interaction, regardless of the boundary condition type.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Enter a value between 0 and 1 for the **Solar Transmissivity Factor**. This will allow you to control the amount of solar irradiation entering the domain. By reducing the solar transmissivity factor from 1 to 0.5, you can effectively cut the total internal energy source entering the domain by half.

> **Important**
>
> Note that the solar transmissivity factor is applied to both direct and diffuse solar irradiation components.

d. Click **OK**.

2. Set the boundary condition for each wall boundary zone that you want to include in solar loading.

   a. Open a **Wall** boundary condition dialog box and click the **Radiation** tab.

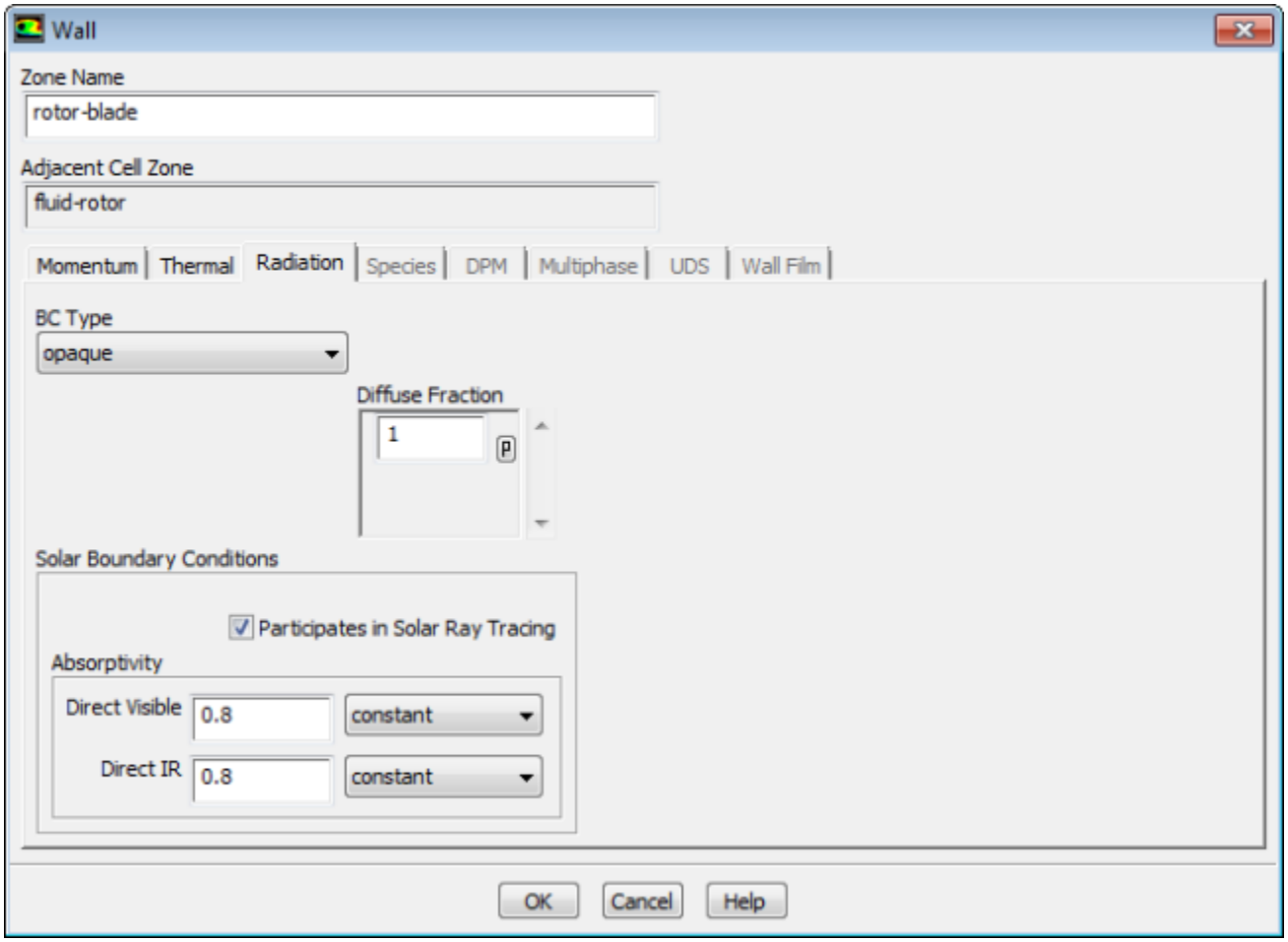
   b. Define the wall as **opaque** or **semi-transparent**. (An opaque wall will not allow any solar radiation to pass through it, while a semi-transparent surface will allow a portion of the solar radiation to pass through it.)

      i. For an opaque wall, select **opaque** from the drop-down list for **BC Type** (Figure 12.37: The Wall Dialog Box (p. 902)). Then enable the **Participates in Solar Ray Tracing** option (this option is enabled for all boundary conditions by default) in the **Solar Boundary Conditions** group box and enter constant values for **Direct Visible** and **Direct IR** absorptivity. Note that if you deactivate solar ray tracing by disabling the **Participates in Solar Ray Tracing** option, the surface will be ignored and the solar ray will pass through it with no interaction, regardless of the boundary condition type.

> **Important**
>
> Absorption in the visible and infrared portions of the spectrum define the surface material for the opaque wall.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

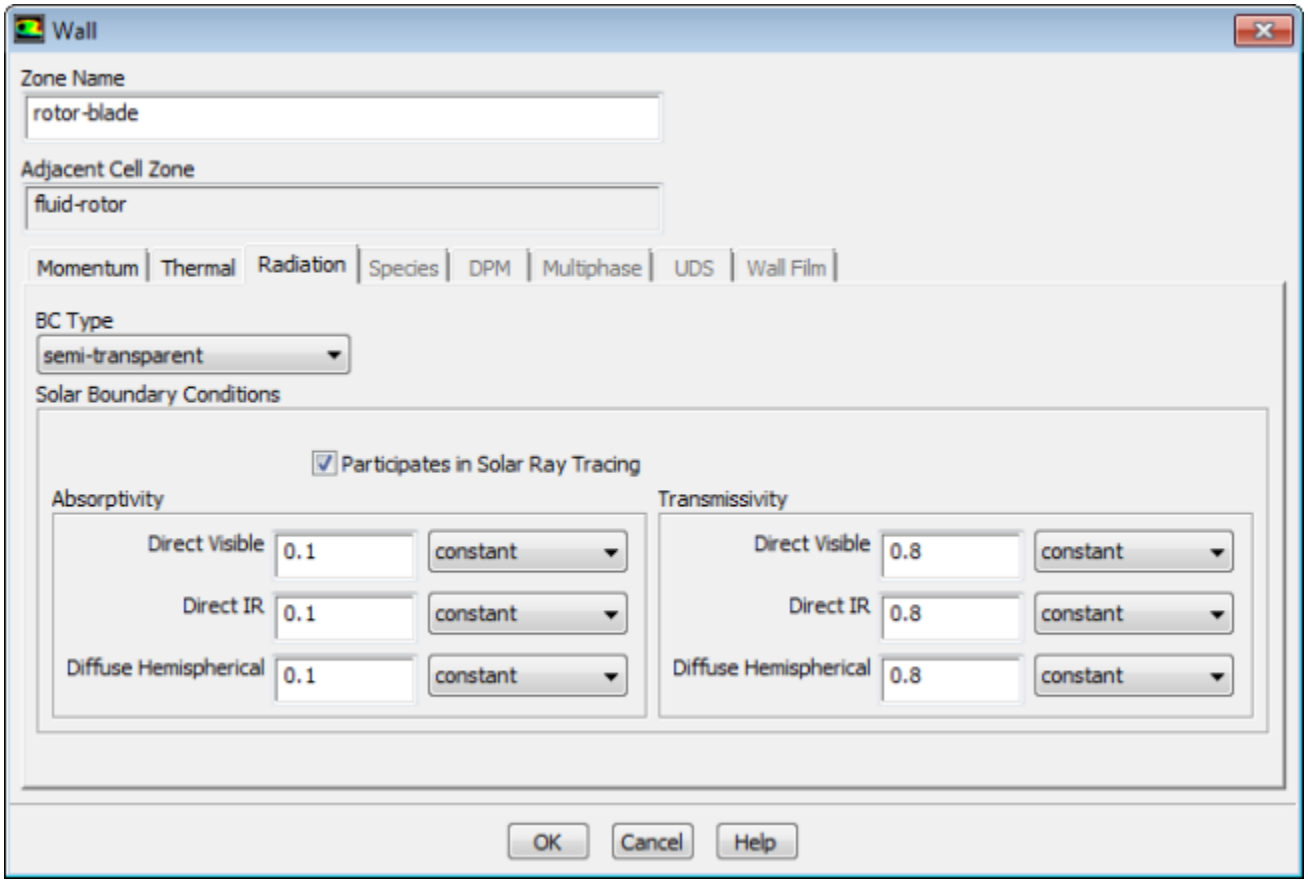
Figure 12.37: The Wall Dialog Box



ii. For a semi-transparent wall, select **semi-transparent** from the drop-down list for **BC Type** (Figure 12.38: The Wall Dialog Box (p. 903)). Then, enable the **Participates in Solar Ray Tracing** option (this option is enabled for all boundary conditions by default) in the **Solar Boundary Conditions** group box and enter constant values for **Direct Visible**, **Direct IR**, and **Diffuse Hemispherical** absorptivity and transmissivity. Note that if you deactivate solar ray tracing by disabling the **Participates in Solar Ray Tracing** option, the surface will be ignored and the solar ray will pass through it with no interaction, regardless of the boundary condition type.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 12.38: The Wall Dialog Box



Important

For semitransparent coupled walls, the solar ray tracing settings are defined on the original wall and not on the shadow wall.

Absorption and transmittance in the visible and infrared portions of the spectrum, as well as the "shading" formulation (**Diffuse Hemispherical**), define the surface material for a semi-transparent wall. These parameters are properties of the glazed unit and should be provided by the glazing manufacturer. The direct components are based on normal incident radiation (ANSYS Fluent adjusts this for the actual angle of incidence). Most manufacturers present this information in a slightly different way so it may be necessary to seek guidance from the supplier. Another useful source of data can be found in the ASHRAE Fundamentals Handbook, chapter on Fenestration.

iii. Click **OK**.

Important

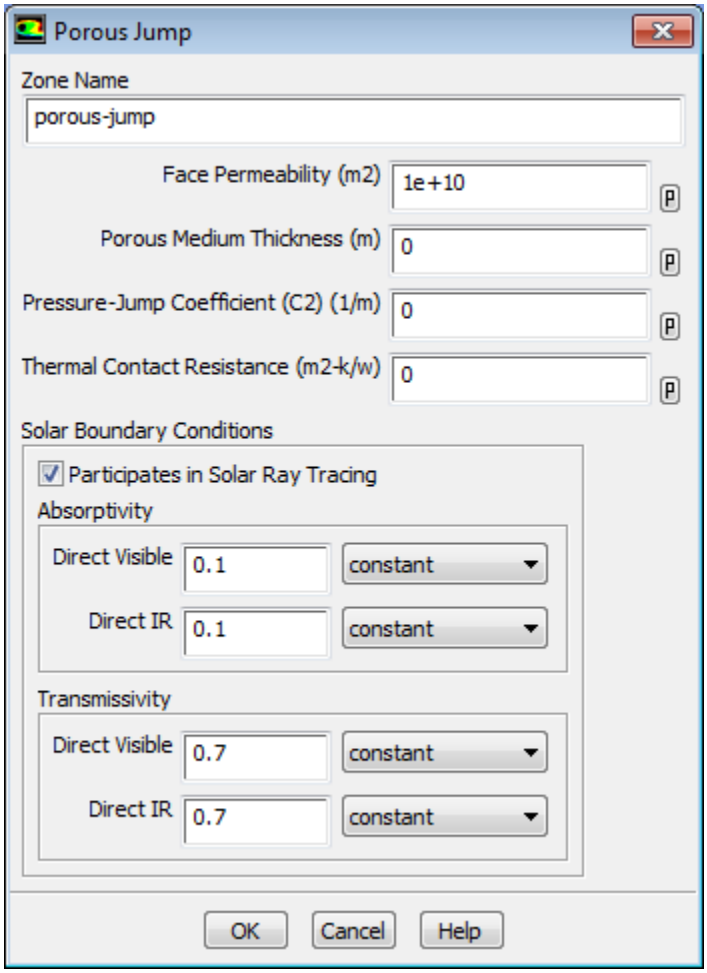
ANSYS Fluent will calculate the reflectivity as the difference between one and the sum of absorptivity and transmissivity:

$$reflectivity = 1 - (absorptivity + transmissivity) \tag{12.21}$$

3. Set the boundary condition for each porous jump boundary zone that you want to include in solar loading. You can define the boundary such that it acts as a semi-transparent surface that will allow a portion of the solar radiation to pass through it, along with fluid flow. In this way you can represent a partial opening (for example, a grate, a grill, or a louver) that is a combination of gaps and (typically) opaque surfaces.

   a. Open the **Porous Jump** boundary condition dialog box (Figure 12.39: The Porous Jump Dialog Box (p. 904)).

   Figure 12.39: The Porous Jump Dialog Box



   b. Define the **Face Permeability**, **Porous Medium Thickness**, **Pressure-Jump Coefficient**, and **Thermal Contact Resistance** as described in User Inputs for the Porous Jump Model (p. 404).

   c. Define the settings in the **Solar Boundary Conditions** group box.

      i. Enable the **Participates in Solar Ray Tracing** option (this option is enabled for all boundary conditions by default). Note that if you deactivate solar ray tracing by disabling the **Participates in Solar Ray Tracing** option, the surface will be ignored and the solar ray will pass through it with no interaction, regardless of the boundary condition type.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ii. Enter constant values (between 0 and 1) for **Direct Visible** and **Direct IR** in the **Absorptivity** group box. These values act as multipliers for the visible and infrared portions of the direct solar radiation spectrum, respectively, to account for the absorption of the porous jump.

> **Important**
>
> One reasonable way to estimate the **Direct Visible** and **Direct IR** absorptivity values for a grill, and so on, is to use the product of the obstructed area fraction and the surface absorptivity. For example, if 40% of the grill facial area is obstructed with grill slats, and the slats have a surface absorptivity of 0.7, you could estimate the absorptivity as being 0.28.

iii. Enter constant values (between 0 and 1) for **Direct Visible** and **Direct IR** in the **Transmissivity** group box. These values act as multipliers for the visible and infrared portions of the direct solar radiation spectrum, respectively, to account for the transmissivity of the porous jump.

> **Important**
>
> One reasonable way to estimate the **Direct Visible** and **Direct IR** transmissivity values for a grill, and so on, is to use the open area fraction. For example, if a grill has openings between the slats that amount to 60% of the area, you could estimate the transmissivity as being 0.6.

d. Click **OK**.

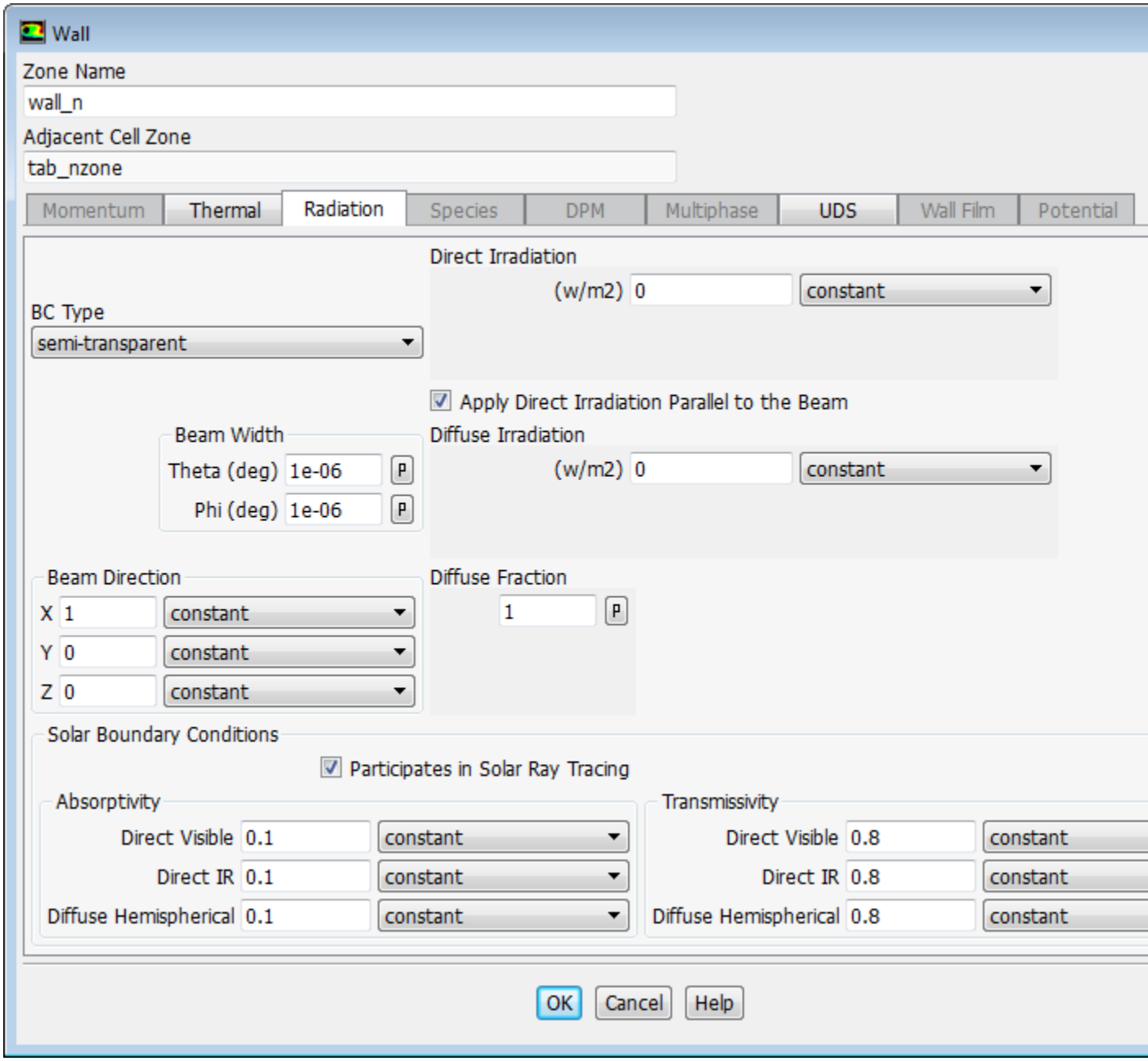
12.3.11.5.5. DO Irradiation

1. For DO irradiation, all boundary conditions are set up as normal for the DO model, except that now you can select semi-transparent boundary surfaces that will provide a source of solar irradiation.

   a. Open a **Wall** boundary condition dialog box and click the **Radiation** tab (Figure 12.40: The Wall Dialog Box (p. 906)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 12.40: The Wall Dialog Box



b. Select **semi-transparent** from the drop-down list for **BC Type**.

c. Enable the **Use Beam Direction from Solar Load Model Settings** option, under **Solar BC Options**, to have the values for beam direction applied from the **Solar Load Model** settings in the **Radiation** dialog box.

Important

Note that the sign of the beam direction that is needed for the DO model is opposite the sun direction vector that is entered or derived from the solar parameters. The beam direction in the DO model is the direction of external radiation (for example, radiation coming from the sun), while the sun direction vector in the solar load

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

model points to the sun. Incident radiation and sun angle always have an opposite sign since they are quantities that are defined from opposite perspectives.

d. Enable the **Use Direct and Diffuse Irradiation from Solar Load Model Settings** option to have the solar calculator output be applied for direct and diffuse irradiation.

When **Use Direct and Diffuse Irradiation from Solar Load Model Settings** is enabled, the beam width will automatically be set to 0.53 degrees - the angle subtended by the sun.

e. Click **OK**.

12.3.11.5.6. Text Interface-Only Commands

ANSYS Fluent has provided some additional commands for solar load setup that are only available in the text interface. These commands are present in the following sections.

12.3.11.5.6.1. Automatically Saving Solar Ray Tracing Data

It is possible to direct ANSYS Fluent to automatically save solar load data to a generic file that you can examine or use in an external program. This is done by executing the text command `autosave-solar-data` from the text interface.

`define` → `models` → `radiation` → `solar-parameters` → `autosave-solar-data`

1. Enter a value for the autosave solar data file frequency when prompted, in order to specify the frequency (in time steps) at which you want the solar load data written to a file. The default value is zero, which means that no automatic saving is performed.

2. Enter the filename, in quotations.

3. Choose to write file in binary format.

The text interface command for `autosave-solar-data` for a file named `solar` and a frequency of `1` is shown below:

```
/define/models/radiation/solar-parameters> autosave-solar-data
 Autosave Solar Data File Frequency [0] 1
 Enter Filename [""] "solar"
```

12.3.11.5.6.2. Automatically Reading Solar Data

When you are executing a transient simulation in parallel ANSYS Fluent and you want to take solar loading conditions into consideration, you can use the `autoread-solar-data` text command to automatically read the solar load data file you generated during a serial run into parallel ANSYS Fluent. This is done by executing the text command `autoread-solar-data` from the text interface.

`define` → `models` → `radiation` → `solar-parameters` → `autoread-solar-data`

1. Enter a value for the autoread solar data file frequency when prompted, in order to specify the frequency (in time steps) at which you want the solar load data read from the file generated during the serial run. The default value is zero, which means that no automatic reading is performed.

2. Enter the filename, in quotations.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The text interface command for autoread-solar-data for a file named solar and a frequency of 1 is shown below:

```
/define/models/radiation/solar-parameters> autoread-solar-data
 Autoread Solar Data File Frequency [0] 1
 Enter Filename [""] "solar"
 Use Binary Format for Reading Data Files [yes]
```

12.3.11.5.6.3. Aligning the Camera Direction With the Position of the Sun

When the solar load model is enabled, you can direct ANSYS Fluent to align the camera direction with the sun position using the text interface command:

define → models → radiation → solar-parameters → sol-camera-pos

This command is useful when you are executing a transient simulation and you want to capture an image of your model with solar load parameters displayed (such as solar heat flux) as the sun position changes with time in order to create an animation. See Postprocessing Solar Load Quantities (p. 909) for details.

12.3.11.5.6.4. Specifying the Scattering Fraction

You can modify the default scattering fraction (1) using the text interface command:

define → models → radiation → solar-parameters → scattering-fraction

The scattering fraction is the amount of direct radiation that has been reflected from opaque surfaces (after entering through the transparent surfaces) that will be considered to remain within the space and be evenly distributed among all surfaces. The value is between 0 and 1.

The text interface command for specifying a scattering-fraction of 0.5 is shown below:

```
/define/models/radiation/solar-parameters> scattering-fraction
 Scattering Fraction [1] .5
```

12.3.11.5.6.5. Applying the Solar Load on Adjacent Fluid Cells

You can direct ANSYS Fluent to apply the solar load that is computed from the solar ray tracing algorithm to adjacent fluid cells by issuing the following command at the text interface:

define → models → radiation → solar-parameters → sol-adjacent-fluidcells

The text interface command is shown below:

```
/define/models/radiation/solar-parameters> sol-adjacent-fluidcells
 Apply Solar Load on adjacent Fluid Cells? [no] y
```

This command allows you to apply solar loads to adjacent fluid cells only, even if solid or shell conduction zones are present. By applying the solar load on adjacent fluid cells, you are overruling the default order of the adjacent cell assignment in ANSYS Fluent which is shell, solid, fluid.

12.3.11.5.6.6. Specifying Quad Tree Refinement Factor

You can modify the default value (7) for the maximum quad tree refinement factor in the solar ray tracing algorithm using the text command:

define → models → radiation → solar-parameters → quad-tree-parameters

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The text interface command is shown below, when a new maximum refinement value of 10 is specified:

```
/define/models/radiation/solar-parameters> quad-tree-parameters
 Maximum Quad-Tree Refinement [7] 10
```

12.3.11.5.6.7. Specifying Ground Reflectivity

You can modify the default value (0.2) for the ground reflectivity using the text command:

`define` → `models` → `radiation` → `solar-parameters` → `ground-reflectivity`

Ground reflectivity ρ_g (Equation 12.19 (p. 893)) includes the contribution of reflected solar radiation from ground surfaces. It is treated as part of the total diffuse solar irradiation when the solar calculator is used in conjunction with the `Diffuse Solar Irradiation` illumination parameter. The default value is 0.2.

```
/define/models/radiation/solar-parameters> ground-reflectivity
 Ground Reflectivity [0.2] 0.5
```

12.3.11.5.6.8. Additional Text Interface Commands

Some solar load commands that are available in the graphical user interface are also made available in the text interface. For example, you can turn the solar load model on using the text command:

`define` → `models` → `radiation` → `solar?`

You can also enter the solar calculator parameters in the text interface by executing the command:

`define` → `models` → `radiation` → `solar-calculator`

Once invoked, you will be prompted to enter the solar calculator input parameters.

To set the illumination parameters, select this option from the `solar-parameters` menu:

`define` → `models` → `radiation` → `solar-parameters` → `illumination-parameters`

And finally, you can direct ANSYS Fluent to compute the solar load on demand, by issuing the text command:

`define` → `models` → `radiation` → `solar-parameters` → `sol-on-demand`

When the command is initiated, the solar data are written to the console.

12.3.11.6. Postprocessing Solar Load Quantities

The following solar load quantities can be used to visualize the illuminated areas and shadows created by solar radiation.

- solar heat flux (that is, sum of visible and IR absorbed solar flux on opaque walls)
- absorbed visible and IR solar flux (semi-transparent walls and porous jump boundaries only)
- reflected visible and IR solar flux (semi-transparent walls and porous jump boundaries only)
- transmitted visible and IR solar flux (semi-transparent walls and porous jump boundaries only)

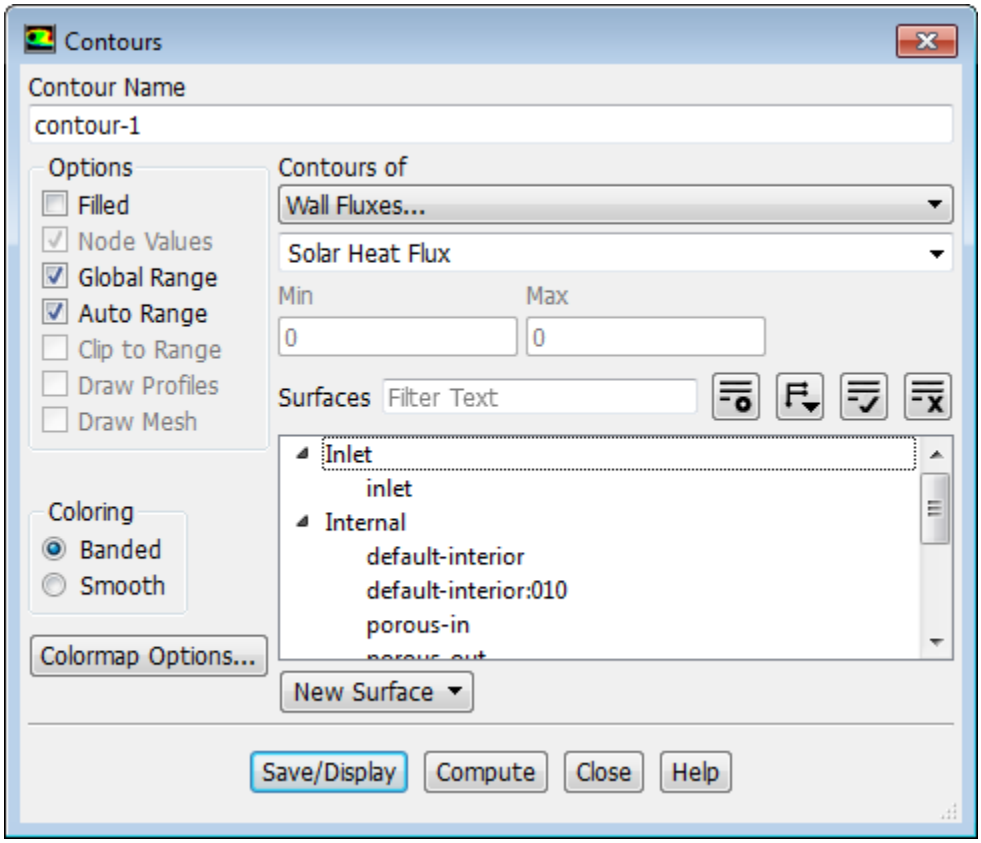
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These quantities are available for postprocessing of solar loading at wall boundaries and can be displayed as contours of **Wall Fluxes** in the **Contours** dialog box. For steady-state simulations, the solar flux data is computed at solution initialization and is available for postprocessing. You can also compute the solar load at any time during your ANSYS Fluent session, after you have set up the model and applied boundary conditions. To compute the solar load on demand, you can issue the `sol-on-demand` command in the text interface (see Additional Text Interface Commands (p. 909) for details).

Solar heat flux, for example, can be displayed for surfaces using the **Contours** dialog box. A sample dialog box is shown below (Figure 12.41: The Contours Dialog Box (p. 910)).

Results → Graphics → Contours New...

Figure 12.41: The Contours Dialog Box



12.3.11.6.1. Solar Load Animation at Different Sun Positions

The solar camera alignment command is useful when you want to take timed pictures of solar loading effects of your model during transient simulations, and later create animations of the image files using an external program. Follow the procedure below.

1. Read (or set up) your transient case file in ANSYS Fluent.

2. Set up the automatic execution of solution commands in the **Execute Commands** dialog box that will: 1) display solar load parameter graphics, 2) re-position the solar camera such that the view is aligned with the instantaneous sun direction, and 3) generate a picture image file (`.tiff`) during the solution process in the **Execute Commands** dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution → **Calculation Activities** → **Execute Commands** **Edit...**

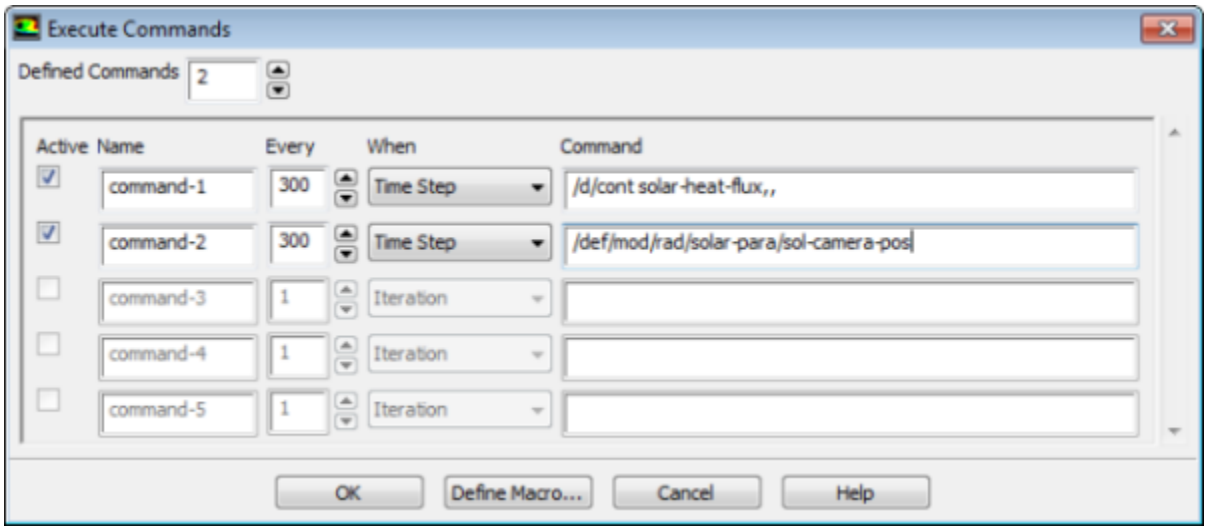
3. Initialize and run the solution.

4. Animate the `.tiff` files using an external animation tool.

The following commands entered in the **Execute Commands** dialog box will direct ANSYS Fluent to display contours of solar heat flux, align the camera with the current direction of the sun, and then generate a picture image file (`.tiff`) of the solar heat flux contour every 300 time steps during the unsteady simulation. See Figure 12.42: The Execute Commands Dialog Box (p. 911).

```
/di/cont solar-heat-flux ,,
/def/mod/rad/solar-para/sol-camera-pos
/di/hc "flux-%t.tiff"
```

Figure 12.42: The Execute Commands Dialog Box



12.3.11.6.2. Reporting and Displaying Solar Load Quantities

ANSYS Fluent provides some additional solar load variables that you can use for postprocessing when your model includes solar ray tracing. You can generate graphical plots or alphanumeric reports of the following variables:

In the **Wall Fluxes...** category:

- **Solar Heat Flux**
- **Transmitted Visible Solar Flux** (semi-transparent walls and porous jump boundaries)
- **Transmitted IR Solar Flux** (semi-transparent walls and porous jump boundaries)
- **Reflected Visible Solar Flux** (semi-transparent walls and porous jump boundaries)
- **Reflected IR Solar Flux** (semi-transparent walls and porous jump boundaries)
- **Absorbed Visible Solar Flux** (semi-transparent walls and porous jump boundaries)
- **Absorbed IR Solar Flux** (semi-transparent walls and porous jump boundaries)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See Field Function Definitions (p. 1931) for their definitions.

12.4. Modeling Periodic Heat Transfer

ANSYS Fluent is able to predict heat transfer in periodically repeating geometries, such as compact heat exchangers, by including only a single periodic module for analysis.

This section discusses streamwise-periodic heat transfer. The treatment of streamwise-periodic flows is discussed in Periodic Flows (p. 582), and a description of no-pressure-drop periodic flow is provided in Periodic Boundary Conditions (p. 386).

Information about streamwise-periodic heat transfer is presented in the following sections:

12.4.1. Overview and Limitations
12.4.2. Theory
12.4.3. Using Periodic Heat Transfer
12.4.4. Solution Strategies for Periodic Heat Transfer
12.4.5. Monitoring Convergence
12.4.6. Postprocessing for Periodic Heat Transfer

12.4.1. Overview and Limitations

The following sections contain information about periodic heat transfer:

12.4.1.1. Overview
12.4.1.2. Constraints for Periodic Heat Transfer Predictions

12.4.1.1. Overview

As discussed in Overview and Limitations (p. 582), streamwise-periodic flow conditions exist when the flow pattern repeats over some length L, with a constant pressure drop across each repeating module along the streamwise direction.

Periodic thermal conditions may be established when the thermal boundary conditions are of the constant wall temperature or wall heat flux type. In such problems, the temperature field (when scaled in an appropriate manner) is periodically fully developed [81] (p. 2891). As for periodic flows, such problems can be analyzed by restricting the numerical model to a single module or periodic length.

12.4.1.2. Constraints for Periodic Heat Transfer Predictions

In addition to the constraints for streamwise-periodic flow discussed in Limitations for Modeling Streamwise-Periodic Flow (p. 583), the following constraints must be met when periodic heat transfer is to be considered:

- The pressure-based solver must be used.
- The thermal boundary conditions must be of the specified heat flux or constant wall temperature type. Furthermore, in a given problem, these thermal boundary types cannot be combined: all boundaries must be either constant temperature or specified heat flux. You can, however, include constant-temperature walls and zero-heat-flux walls in the same problem. For the constant-temperature case, all walls must be at the same temperature (profiles are not allowed) or zero heat flux. For the heat flux case, profiles and/or different values of heat flux may be specified at different walls.
- When constant-temperature wall boundaries are used, you cannot include viscous heating effects or any volumetric heat sources.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- In cases that involve solid regions, the regions cannot straddle the periodic plane.
- The thermodynamic and transport properties of the fluid (heat capacity, thermal conductivity, viscosity, and density) cannot be functions of temperature. (You cannot, therefore, model reacting flows.) Transport properties may, however, vary spatially in a periodic manner, and this allows you to model periodic turbulent flows in which the effective turbulent transport properties (effective conductivity, effective viscosity) vary with the (periodic) turbulence field.

Theory (p. 913) and Using Periodic Heat Transfer (p. 914) provide more detailed descriptions of the input requirements for periodic heat transfer.

12.4.2. Theory

Streamwise-periodic flow with heat transfer from constant-temperature walls is one of two classes of periodic heat transfer that can be modeled by ANSYS Fluent. A periodic fully developed temperature field can also be obtained when heat flux conditions are specified. In such cases, the temperature change between periodic boundaries becomes constant and can be related to the net heat addition from the boundaries as described in this section.

Important

Periodic heat transfer can be modeled only if you are using the pressure-based solver.

12.4.2.1. Definition of the Periodic Temperature for Constant- Temperature Wall Conditions

For the case of constant wall temperature, as the fluid flows through the periodic domain, its temperature approaches that of the wall boundaries. However, the temperature can be scaled in such a way that it behaves in a periodic manner. A suitable scaling of the temperature for periodic flows with constant-temperature walls is [81] (p. 2891)

$$\theta = \frac{T(\vec{r}) - T_{wall}}{T_{bulk,inlet} - T_{wall}} \tag{12.22}$$

The bulk temperature, $T_{bulk,inlet}$, is defined by

$$T_{bulk,inlet} = \frac{\int_A T \left| \rho \vec{v} \cdot d\vec{A} \right|}{\int_A \left| \rho \vec{v} \cdot d\vec{A} \right|} \tag{12.23}$$

where the integral is taken over the inlet periodic boundary (A). It is the scaled temperature, θ, which obeys a periodic condition across the domain of length L.

12.4.2.2. Definition of the Periodic Temperature Change σ for Specified Heat Flux Conditions

When periodic heat transfer with heat flux conditions is considered, the form of the unscaled temperature field becomes analogous to that of the pressure field in a periodic flow:

$$\frac{T(\vec{r}+\vec{L}) - T(\vec{r})}{L} = \frac{T(\vec{r}+2\vec{L}) - T(\vec{r}+\vec{L})}{L} = \sigma. \tag{12.24}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where $\vec{L}$ is the periodic length vector of the domain. This temperature gradient, σ, can be written in terms of the total heat addition within the domain, Q, as

$$\sigma = \frac{Q}{\dot{m} c_p L} = \frac{T_{bulk,exit} - T_{bulk,inlet}}{L} \tag{12.25}$$

where $\dot{m}$ is the specified or calculated mass flow rate.

12.4.3. Using Periodic Heat Transfer

A typical calculation involving both streamwise-periodic flow and periodic heat transfer is performed in two parts. First, the periodic velocity field is calculated (to convergence) without consideration of the temperature field. Next, the velocity field is frozen and the resulting temperature field is calculated. These periodic flow calculations are accomplished using the following procedure:

1. Set up a mesh with translationally periodic boundary conditions.
2. Specify constant thermodynamic and molecular transport properties.
3. Specify either the periodic pressure gradient or the net mass flow rate through the periodic boundaries.
4. Compute the periodic flow field, solving momentum, continuity, and (optionally) turbulence equations.
5. Specify the thermal boundary conditions at walls as either heat flux or constant temperature.
6. Define an inlet bulk temperature.
7. Solve the energy equation (only) to predict the periodic temperature field.

In order to model the periodic heat transfer, you will need to set up your periodic model in the manner described in User Inputs for the Pressure-Based Solver (p. 584) for periodic flow models with the pressure-based solver, noting the restrictions discussed in Limitations for Modeling Streamwise-Periodic Flow (p. 583) and Constraints for Periodic Heat Transfer Predictions (p. 912). In addition, you will need to provide the following inputs related to the heat transfer model:

1. Enable the solution of the energy equation by right-clicking **Energy** in the tree (under **Setup/Models**) and clicking **On** in the menu that opens (Figure 12.1: Enabling the Energy Equation (p. 826)).

   Setup → **Models** → **Energy** → **On**

2. Define the thermal boundary conditions according to one of the following procedures. The boundary condition dialog boxes can be opened by right-clicking the boundary name in the tree (under **Setup/Boundary Conditions**) and clicking **Edit...** in the menu that opens; alternatively, you can open them from the **Boundary Conditions** task page:

   Setup → **Boundary Conditions**

   - If you are modeling periodic heat transfer with specified-temperature boundary conditions, set the wall temperature T_{wall} for all wall boundaries in their respective **Wall** dialog box. Note that all wall boundaries must be assigned the same temperature and that the entire domain (except the periodic boundaries) must be "enclosed" by this fixed-temperature condition, or by symmetry or adiabatic (q=0) boundaries.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you are modeling periodic heat transfer with specified-heat-flux boundary conditions, set the wall heat flux in the **Wall** dialog box for each wall boundary. You can define different values of heat flux on different wall boundaries, but you should have no other types of thermal boundary conditions active in the domain.

3. Define solid regions, if appropriate, according to one of the following procedures. The cell zones can be defined by right-clicking the zone name in the tree (under **Setup/Cell Zone Conditions**), or using the **Cell Zone Conditions** task page:

   Setup → **Cell Zone Conditions**

   - If you are modeling periodic heat transfer with specified-temperature conditions, conducting solid regions can be used within the domain, provided that on the perimeter of the domain they are enclosed by the fixed-temperature condition. Heat generation within the solid regions is not allowed when you are solving periodic heat transfer with fixed-temperature conditions.

   - If you are modeling periodic heat transfer with specified-heat-flux conditions, you can define conducting solid regions at any location within the domain, including volumetric heat addition within the solid, if desired.

4. Set constant material properties (density, heat capacity, viscosity, thermal conductivity), *not* temperature-dependent properties, using the **Create/Edit Materials** dialog box. This dialog box can be opened by right-clicking the material name in the tree (under **Setup/Materials**) and clicking **Edit...** in the menu that opens; alternatively, you can open them from the **Materials** task page:

   Setup → **Materials**

5. Specify the **Upstream Bulk Temperature** in the **Periodic Conditions** dialog box, which can be opened from **Boundary Conditions** task page:

   Setup → **Boundary Conditions** → **Periodic Conditions...**

   Important

   If you are modeling periodic heat transfer with specified-temperature conditions, the bulk temperature should not be equal to the wall temperature, since this will give you the trivial solution of constant temperature everywhere.

6. Set the solution parameters as described in Solution Strategies for Periodic Heat Transfer (p. 915).

7. Run the solution and monitor the convergence as described in Monitoring Convergence (p. 916).

8. Postprocess the results as described in Postprocessing for Periodic Heat Transfer (p. 916).

12.4.4. Solution Strategies for Periodic Heat Transfer

After completing the inputs described in Using Periodic Heat Transfer (p. 914), you can solve the flow and heat transfer problem to convergence. The most efficient approach to the solution, however, is a sequential one in which the periodic flow is first solved without heat transfer and then the heat transfer is solved leaving the flow field unaltered. This sequential approach is accomplished as follows:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Disable solution of the energy equation in the **Equations** dialog box; this dialog box is accessed by right-clicking **Solution Controls** in the tree (under **Solution**) and clicking **Equations...** in the menu that opens.

   Solution → **Controls** **Equations...**

2. Solve the remaining equations (continuity, momentum, and, optionally, turbulence parameters) to convergence to obtain the periodic flow field.

   Important

   When you initialize the flow field before beginning the calculation, use the mean value between the inlet bulk temperature and the wall temperature for the initialization of the temperature field.

3. Return to the **Equations** dialog box and turn off solution of the flow equations and turn on the energy solution.

4. Solve the energy equation to convergence to obtain the periodic temperature field of interest.

While you can solve your periodic flow and heat transfer problems by considering both the flow and heat transfer simultaneously, you will find that the procedure outlined above is more efficient.

12.4.5. Monitoring Convergence

If you are modeling periodic heat transfer with specified-temperature conditions, you can monitor the value of the bulk temperature ratio

$$\theta = \frac{T_{wall} - T_{bulk,inlet}}{T_{wall} - T_{bulk,exit}} \tag{12.26}$$

during the calculation by creating a report plot that includes bulk temperature ratio (**per-bulk-temp-ratio**), to ensure that you reach a converged solution. See Monitoring Statistics (p. 1633) for additional information.

12.4.6. Postprocessing for Periodic Heat Transfer

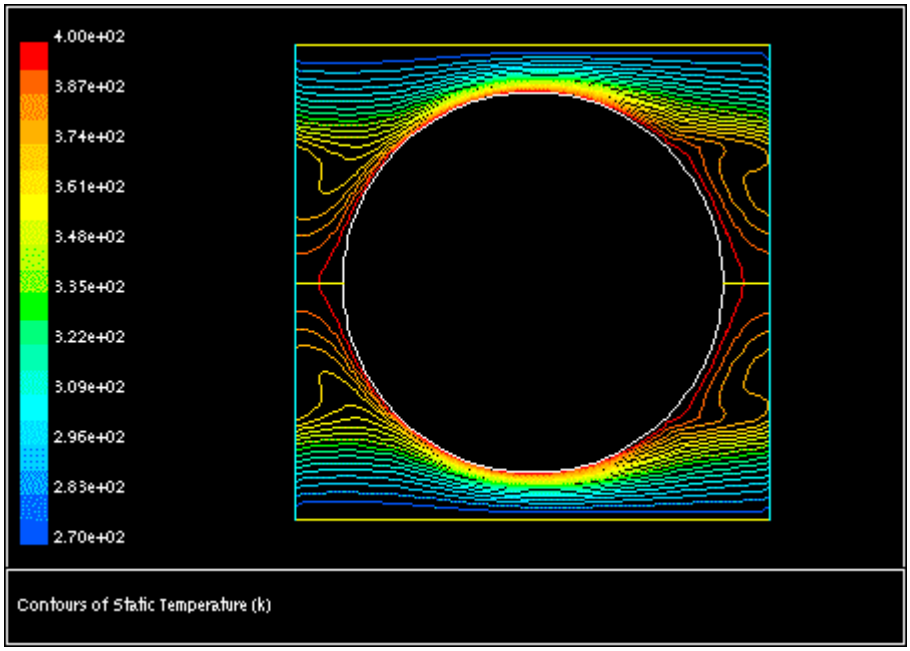
The actual temperature field predicted by ANSYS Fluent in periodic models will not be periodic, and viewing the temperature results during postprocessing will display this actual temperature field ($T(\vec{r})$ of Equation 12.22 (p. 913)). The displayed temperature may exhibit values outside the range defined by the inlet bulk temperature and the wall temperature. This is permissible since the actual temperature profile at the inlet periodic face will have temperatures that are higher or lower than the inlet bulk temperature.

Static Temperature is found in the **Temperature...** category of the variable selection drop-down list that appears in postprocessing dialog boxes.

Figure 12.43: Temperature Field in a 2D Heat Exchanger Geometry With Fixed Temperature Boundary Conditions (p. 917) shows the temperature field in a periodic heat exchanger geometry.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 12.43: Temperature Field in a 2D Heat Exchanger Geometry With Fixed Temperature Boundary Conditions



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

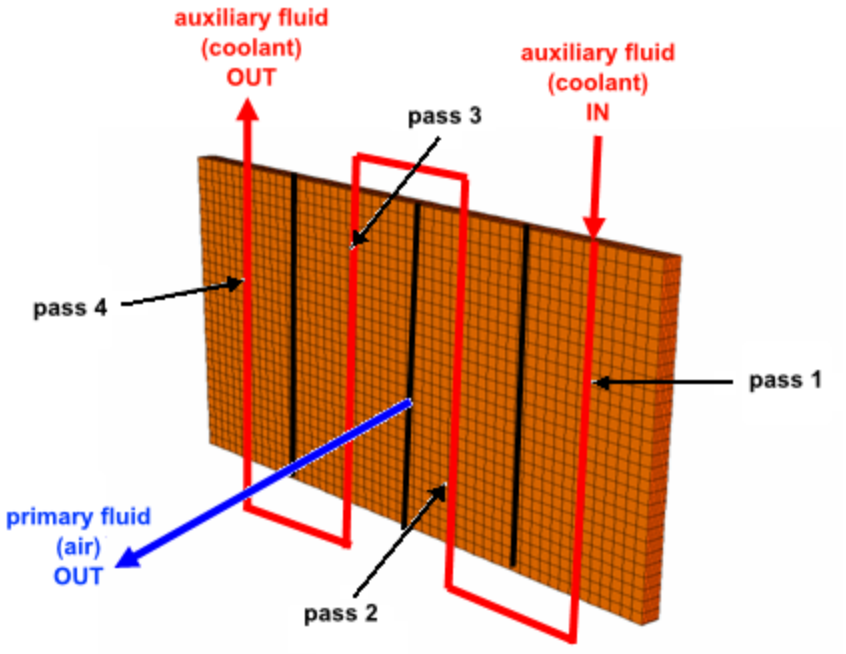
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 13: Modeling Heat Exchangers

Many engineering systems, including power plants, climate control, and engine cooling systems typically contain tubular heat exchangers. However, for most engineering problems, it is impractical to model individual fins and tubes of a heat exchanger core. In principle, heat exchanger cores introduce a pressure drop to the primary fluid stream and transfer heat from or to a second fluid (for example, a coolant), referred to here as the auxiliary fluid.

ANSYS Fluent provides two distinct methods of modeling a heat exchanger: the dual cell model and the macro model. These models can be used to compute the auxiliary fluid inlet temperature for a fixed heat rejection or the total heat rejection for a fixed auxiliary fluid inlet temperature. The dual cell model allows the solution of the passes of the auxiliary flow on a separate mesh, that is, other than the primary fluid mesh (see Figure 13.1: An Example of a Four-Pass Heat Exchanger (p. 919)), unlike the macro model, where the auxiliary flow passes are modeled as 1D flow.

Figure 13.1: An Example of a Four-Pass Heat Exchanger



Important

Note that the heat exchanger models are not appropriate for modeling a cold-only flow; in such a case, you should instead use the porous media model, as described in Porous Media Conditions (p. 261).

For theoretical information about the various heat exchanger models, refer to Heat Exchangers in the Theory Guide.

The following sections contain information about the heat exchanger models:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.1. Choosing a Heat Exchanger Model
13.2. The Dual Cell Model
13.3. The Macro Heat Exchanger Models
13.4. Postprocessing for the Heat Exchanger Model
13.5. Useful Reporting TUI Commands

13.1. Choosing a Heat Exchanger Model

ANSYS Fluent provides various options for modeling heat exchangers, each with their own features and limitations. The following instructions can help you determine which option/combination of options is the most appropriate for your problem.

1. Decide whether you want to use the dual cell model or the macro model.

   a. The dual cell model provides the greatest flexibility with regard to the shape of the heat exchanger core and the nature of the mesh, and allows the auxiliary fluid to be highly non-uniform as it enters the core (for example, due to passing through arbitrary shaped inlet tanks). However, the dual cell model may need to be discounted as an option because of the following limitations:

      - If the heat exchanger performance data that you have is in the form of a velocity vs. effectiveness curve, the dual cell model cannot be used.
      - The dual cell model does not allow you to model phase change in the auxiliary fluid.

      If the previous limitations are not relevant for your problem, you can proceed directly to using the dual cell model, as described in Using the Dual Cell Heat Exchanger Model (p. 922).

   b. While the macro model has more restrictions than the dual cell model, it is quite suitable for a thin 3D heat exchanger core with a rectangular cross-section, where the pass-to-pass plane is perpendicular to the primary flow direction, the auxiliary flow is uniform (so it can be treated as a 1D flow), and the mesh is uniform and structured. To verify that your problem can tolerate the limitations of the macro model, see Restrictions (p. 931).

2. If you chose to use the macro model in the previous step, you must decide whether you want to use the grouped or ungrouped version of this model. If you want to define a single heat exchanger using multiple fluid zones, or if you would like to connect the fluid flow path among multiple heat exchangers, you should use the grouped version, as described in Using the Grouped Macro Heat Exchanger Model (p. 943). Otherwise, you can use the ungrouped version, as described in Using the Ungrouped Macro Heat Exchanger Model (p. 932).

3. If you chose to use the macro model in the step 1., you must decide whether you want to model the heat transfer using the number-of-transfer-units (NTU) method or the simple effectiveness method. This decision largely rests on the kind of experimental data that you have: the NTU method requires that you provide the various primary fluid flow rates and auxiliary fluid flow rates and the corresponding heat transfer values; the simple effectiveness method requires that you provide the data points for a curve that defines how the effectiveness (that is, the ratio of actual rate of heat transfer between the primary and auxiliary fluids to the maximum possible rate of heat transfer)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

varies with the fluid velocity. Additionally, you should consider the differences between the two heat transfer models outlined in Table 13.1: NTU Model Vs. Simple Effectiveness Model (p. 921).

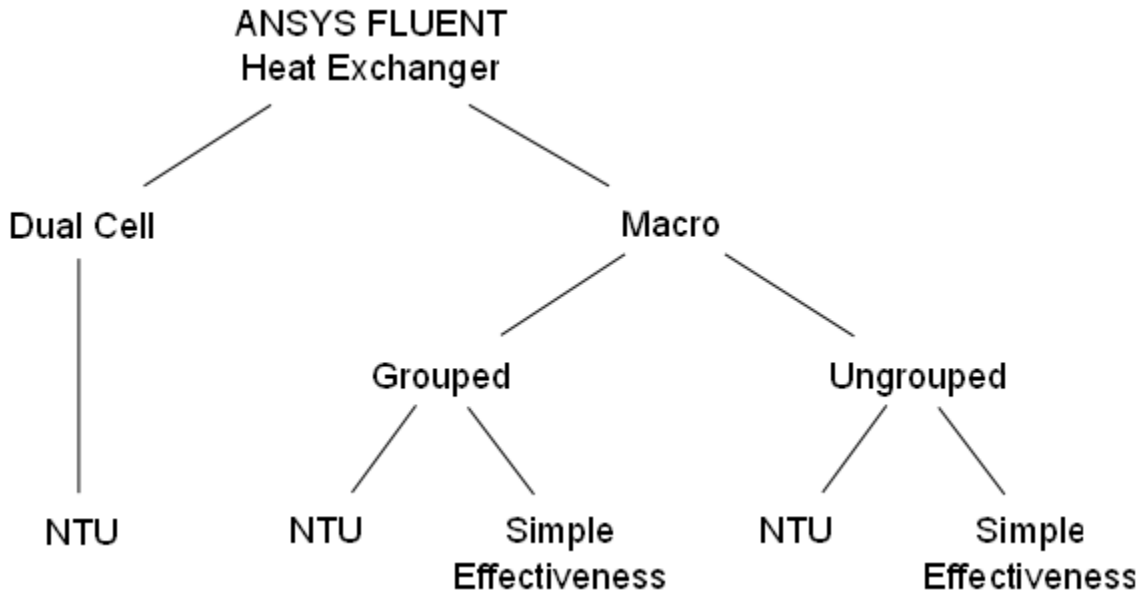
Table 13.1: NTU Model Vs. Simple Effectiveness Model

	NTU Model	Simple Effectiveness Model
How many phases are allowed in the auxiliary flow?	single phase only	single phase and two phase
Can you model primary fluid-side reverse flow?	Yes	No
Can the primary fluid have a variable density?	Yes	No
Must the primary fluid heat capacity be less than the auxiliary fluid heat capacity?	No	Yes

You will specify whether you want the NTU or simple effectiveness model in the **Model Data** tab of either the **Ungrouped Macro Heat Exchanger** dialog box or the **Heat Exchanger Group** dialog box, depending on whether you opted for the ungrouped or grouped version of the macro model, respectively.

An overview of the options available to you when modeling heat exchangers is shown in Figure 13.2: Heat Exchanger Modeling Options (p. 921).

Figure 13.2: Heat Exchanger Modeling Options



13.2. The Dual Cell Model

The dual cell heat exchanger model allows the solution of both the primary and auxiliary flow on separate co-located meshes and couples the two flows only through heat transfer at the heat exchanger core.

For theoretical information about this model, refer to The Dual Cell Model in the Theory Guide.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.2.1. Restrictions

The following restrictions exist for the dual cell heat exchanger models:

- The simple effectiveness model is not available.
- The heat exchanger performance data must be in the form of heat transfer rates or number of transfer units (NTU) for auxiliary / primary fluid flow rate combinations, rather than a velocity vs. effectiveness curve.
- In the case of a heat exchanger in which the primary and auxiliary meshes are not identical, heat transfer may be non-conservative (that is, the heat lost by the hot fluid may not equal the heat gained by the cold fluid). To minimize the difference in heat transfer, the topology and size of the primary and auxiliary cells should be as similar as possible, with the ideal being one-to-one cell conductivity. Note that you can make the meshes identical by copying a zone via the `mesh/modify-zones/copy-move-cell-zone` text command.

13.2.2. Using the Dual Cell Heat Exchanger Model

The steps for setting up the dual cell heat exchanger model is as follows:

1. Read the mesh file.

   a. If the mesh file does not already contain overlapping heat exchanger cores for primary and auxiliary fluids, then you must create a duplicate of the zone that represents the core using the following text command:

      `mesh` → `modify-zones` → `copy-move-cell-zone`

      See Copying Cell Zones (p. 222) for details on copying meshes.

   b. Make sure that the auxiliary fluid mesh is divided into separate zones, one for each pass.

      Setting Up Domain → **Zones** → **Separate** → **Cells...**

      See Separating Cell Zones (p. 210) for details on separating meshes.

   c. Make sure that the inlet and outlet for the heat exchanger are two separate zones.

2. Enable the calculation of energy in the **Energy** dialog box.

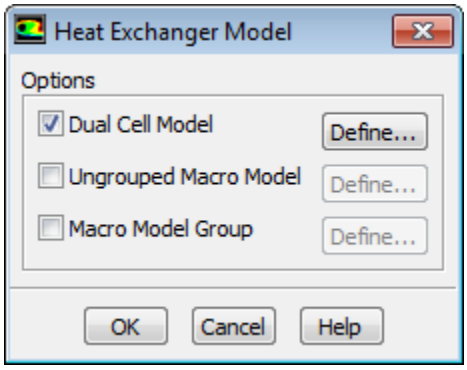
   Setup → **Models** → **Energy** **ON**

3. Enable the **Dual Cell Model** in the **Heat Exchanger Model** dialog box and click **Define...** (Figure 13.3: The Heat Exchanger Model Dialog Box (p. 923)).

   Setup → **Models** → **Heat Exchanger** **Edit...**

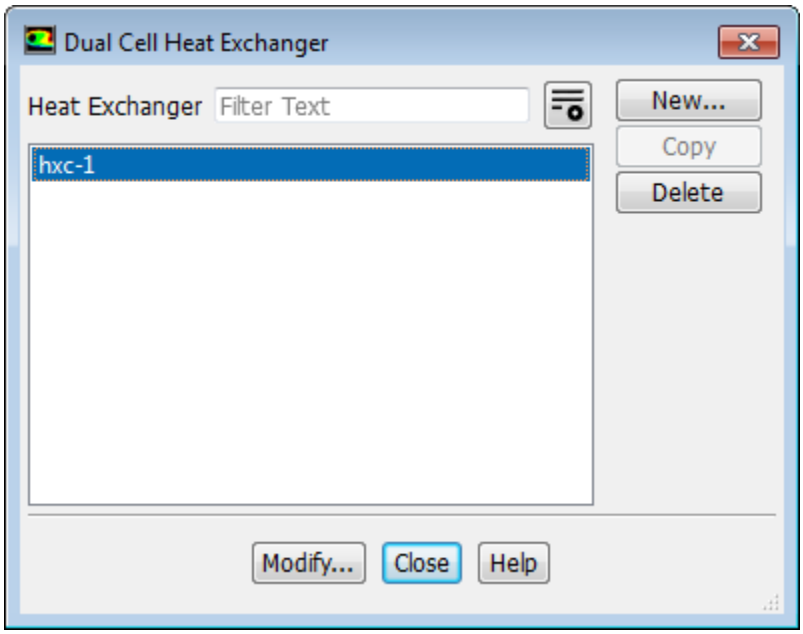
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.3: The Heat Exchanger Model Dialog Box



4. Specify the inputs to the dual cell heat exchanger model, using the **Dual Cell Heat Exchanger** dialog box (Figure 13.4: The Dual Cell Heat Exchanger Dialog Box (p. 923)).

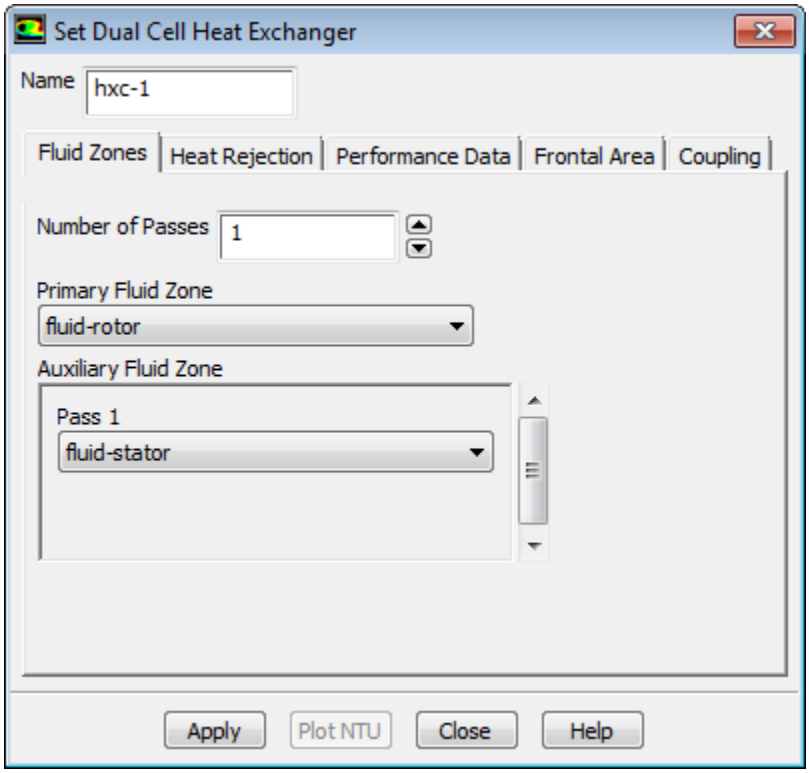
Figure 13.4: The Dual Cell Heat Exchanger Dialog Box



5. Click **New...** to define the heat exchanger. The **Set Dual Cell Heat Exchanger** dialog box will appear (Figure 13.5: The Set Dual Cell Heat Exchanger Dialog Box (p. 924)), where you will define the heat exchanger parameters.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.5: The Set Dual Cell Heat Exchanger Dialog Box



a. Enter the heat exchanger **Name** or keep the default name. The suffix `-1` is incremented automatically on defining more than one heat exchanger.

b. In the **Fluid Zones** tab (Figure 13.5: The Set Dual Cell Heat Exchanger Dialog Box (p. 924))

   i. Specify the **Number of Passes** of your heat exchanger.

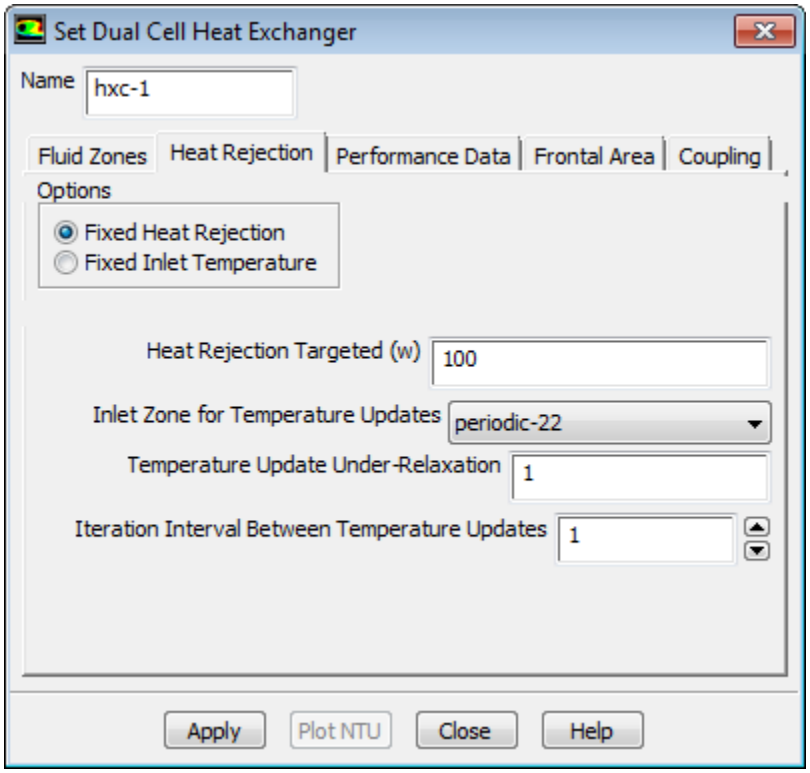
   ii. Select the appropriate **Primary** and **Auxiliary Fluid Zone**, representing the heat exchanger core.

> **Important**
>
> The selected zones must be overlapping in physical space.

c. Click the **Heat Rejection** tab (Figure 13.6: The Heat Rejection Tab (p. 925)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.6: The Heat Rejection Tab



i. If you select **Fixed Heat Rejection**, set the inputs for the following:

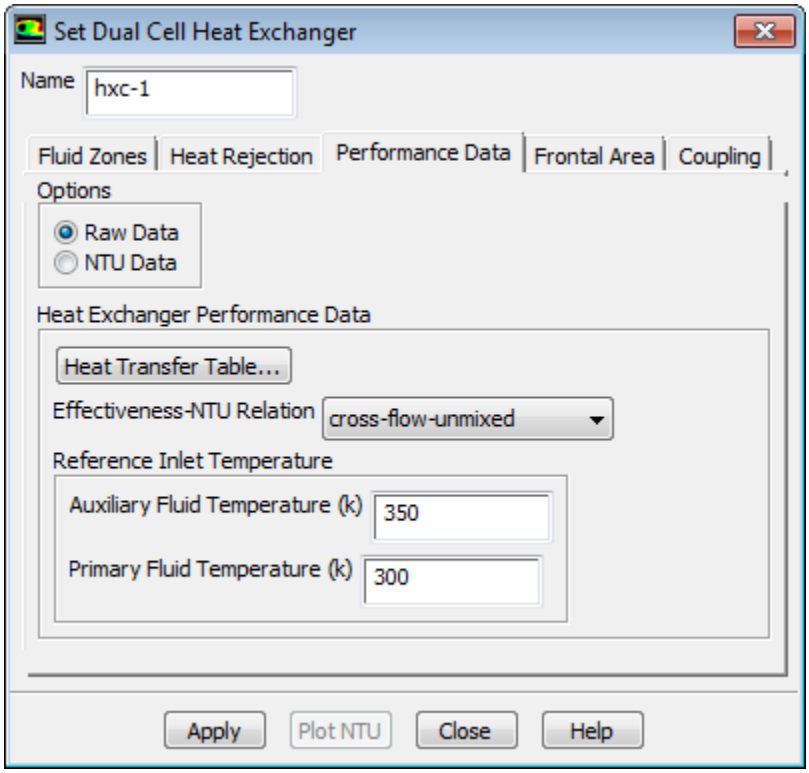
- **Heat Rejection Targeted** which is the heat rejection desired from the heat exchanger.
- **Inlet Zone for Temperature Updates** allows ANSYS Fluent to change the temperature of the specified inlet zone in order to match the targeted heat rejection.
- **Temperature Update Under-Relaxation** is a factor that controls convergence.
- **Iteration Interval Between Temperature Updates** is used to control divergence.

ii. Select **Fixed Inlet Temperature** if the output desired is total heat rejection.

d. Click the **Performance Data** tab (Figure 13.7: The Performance Data Tab (p. 926)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.7: The Performance Data Tab



You must decide whether you will select **Raw Data** or **NTU Data** from the **Options** list. Using the **Raw Data** option specifies that the effectiveness (ε) is evaluated as the ratio of the rate of heat transfer between the primary and auxiliary fluids (which you define in a table format) to the maximum possible rate of heat transfer (q_{max}) for the current flow regime (see Equation 6.7 in the Theory Guide). The calculated efficiency is then used to calculate the NTU value using the crossflow formula for each and every set of primary and auxiliary flow rates.

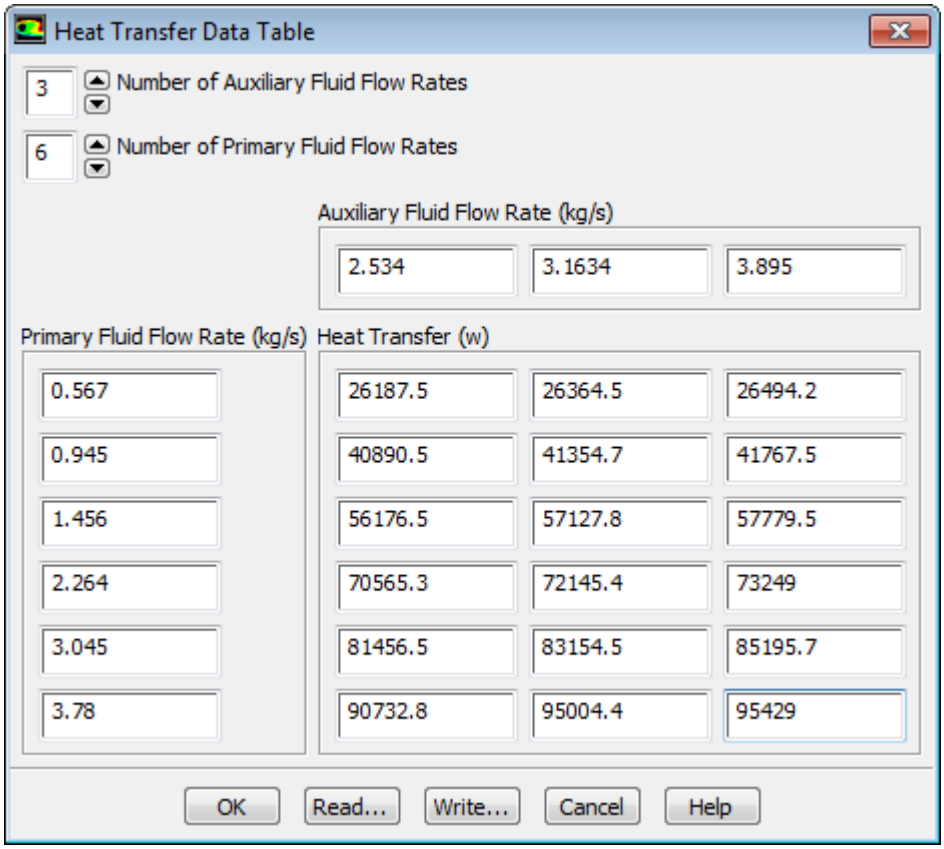
If the heat exchanger manufacturer used a similar approach in evaluating the NTU values, then there is no difference between the **Raw Data** and **NTU Data** options. However, if a different methodology was used by the manufacturer that accounts for some possible non-uniformity of the profile or for the partial mixing of the flow stream, then the **NTU Data** option is likely to be more accurate. Similarly, you may be able to assess more accurate NTU values for your case using tables from the relevant literature. This can even include some mixing of the oncoming flow stream, for situations where the primary flow can deviate from the direction normal to the heat exchanger tubes; this can happen in heat exchangers where the fins are either very spaced or absent. Therefore, the **NTU Data** option can be more accurate if the NTU values are gathered at non-ideal conditions that bear a close resemblance to your simulated scenario.

i. If you select the **Raw Data** option, then specify the following:

- **Heat Transfer Data...** opens the Heat Transfer Data Table Dialog Box (p. 2204), where you will enter experimental data that defines how heat transfer values relate to the fluid flow rates (Figure 13.8: The Heat Transfer Data Table Dialog Box (p. 927)). See Specifying Heat Exchanger Performance Data (p. 937) more details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.8: The Heat Transfer Data Table Dialog Box



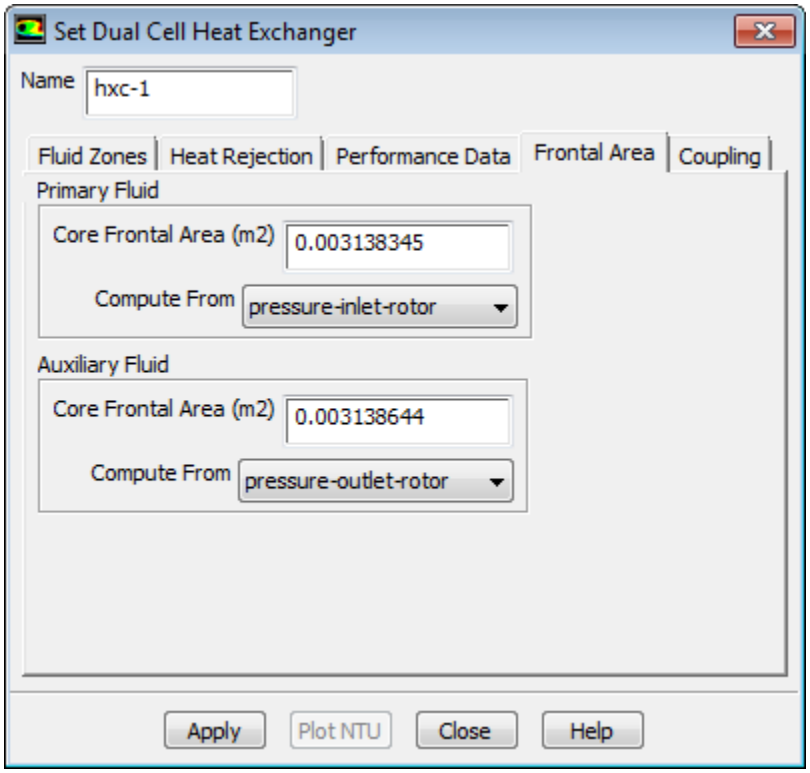
- **Effectiveness-NTU Relation** computes the NTU values from the heat transfer data. Choose **cross-flow-unmixed**, **parallel-flow**, or **counter-flow**, all of which are described in NTU Relations.
- **Auxiliary Fluid Temperature** is the inlet reference temperature for the auxiliary fluid.
- **Primary Fluid Temperature** is the inlet reference temperature for the primary fluid.

ii. If you select the **NTU Data** option, click **NTU Table...** to access the **NTU Table** dialog box. Populate this table in the same manner as described previously for the Heat Transfer Data Table Dialog Box (p. 2204). More information is available in Specifying Heat Exchanger Performance Data (p. 937).

e. Click the **Frontal Area** tab. You have the option to specify the **Primary** and **Auxiliary Fluid Core Frontal Area** directly, or compute the area from a surface zone, as shown in Figure 13.9: The Frontal Area Tab (p. 928).

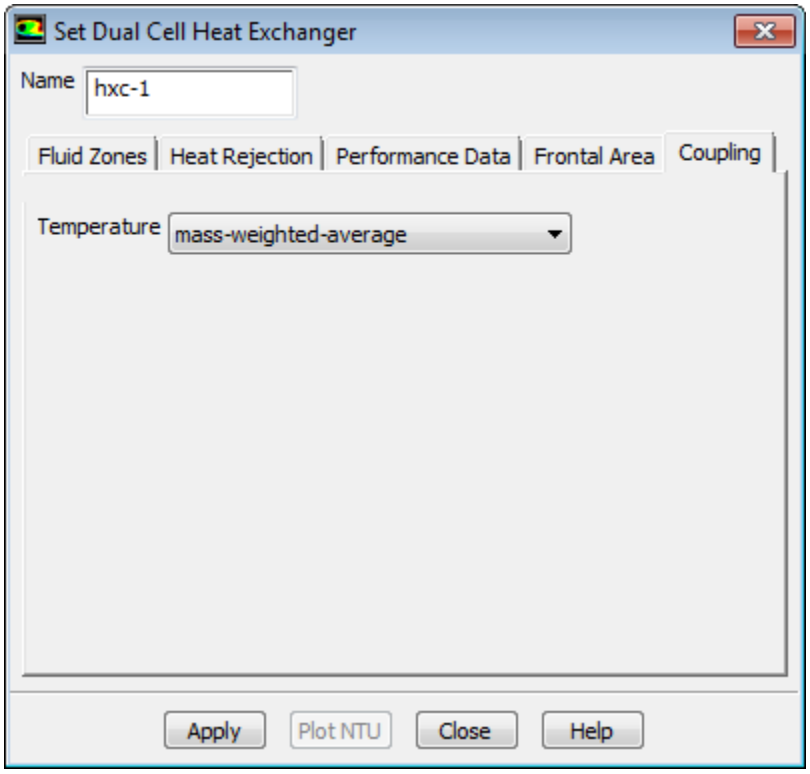
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.9: The Frontal Area Tab



f. Click the **Coupling** tab if you want to couple the heat exchanger passes (Figure 13.10: The Coupling Tab (p. 928)).

Figure 13.10: The Coupling Tab



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Consider the following example illustrating the coupling of a four-pass heat exchanger.

Figure 13.11: An Example of a Four-Pass Heat Exchanger

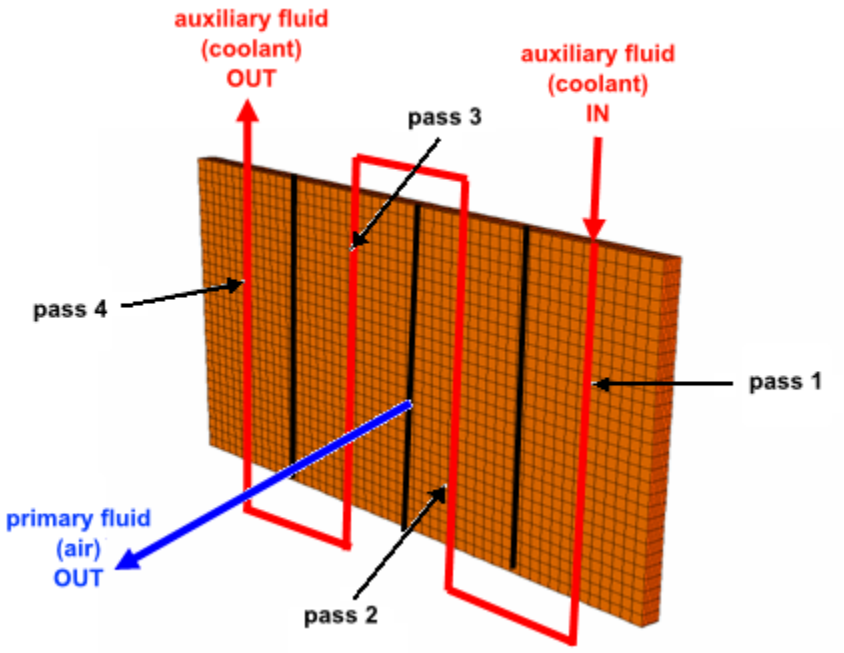


Figure 13.11: An Example of a Four-Pass Heat Exchanger (p. 929) shows a four-pass heat exchanger with air as the primary fluid and the coolant as the auxiliary fluid. The coolant flows through the tubes in a serpentine manner and air flows normal to the tubes, forming a cross flow pattern. To model this type of flow using the dual cell heat exchanger model, you must first generate the mesh. The mesh should contain the following:

i. A single primary cell zone.

ii. Four adjacent auxiliary cell zones, one for each pass. Each auxiliary zone should be separated from the other by a coupled or uncoupled wall. Each pass will have its own inlet and outlet zones.

iii. The primary and four auxiliary zones should overlap in physical space.

In the **Coupling** tab, **mass-weighted-average** is selected by default for the **Temperature** of the outlet of Pass 1 to the inlet of Pass 2. Similarly, the mass-weighted-average temperature of the outlet of Pass 2 will be applied at the inlet zone of Pass 3, and so on. Alternatively, you can couple the passes by using **Profiles...** in the **Boundary Conditions** task page or by using user-defined functions to define the **Temperature** in the **Thermal** tab of the relevant boundary condition dialog boxes. If you do so, make sure you select **none** from the **Temperature** drop-down list in the **Coupling** tab of the **Set Dual Heat Exchanger** dialog box.

Important

Make sure to specify the auxiliary zones in the correct order (that is the zone for Pass 1 should be selected first, then Pass 2, and so on) in the **Fluid Zones** tab of the **Set Dual Cell Heat Exchanger** dialog box.

g. Click **Apply** to save the heat exchanger inputs.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. To view the plot of NTU vs. primary mass flow rate for each auxiliary mass flow rate, click **Plot NTU**.

   The **Plot NTU** button will plot the performance data curve for the selected heat exchanger. The performance data is supplied through the **Performance Data** tab.

   When you close the **Set Dual Cell Heat Exchanger** dialog box, you will return to the **Dual Cell Heat Exchanger** dialog box, where you should now see the heat exchanger name in the **Heat Exchanger** list.

   You can

   - Modify the settings of heat exchanger by selecting it from the list and clicking **Modify...**.
   - Copy the data of one heat exchanger to another using the **Copy** button, assuming you have more than one heat exchanger.
   - Delete any unwanted heat exchangers by selecting the heat exchanger from the list and clicking **Delete**.

   Important

   All the inputs are copied except for the name, primary fluid zone and auxiliary fluid zone.

13.3. The Macro Heat Exchanger Models

To use the macro heat exchanger model, you must define one or more fluid zone(s) to represent the heat exchanger core. Typically, the fluid zone is sized to the dimension of the core itself. As part of the setup procedure, you will define the auxiliary fluid path, the number of macros, and the physical properties and operating conditions of the core (pressure drop parameters, heat exchanger effectiveness, auxiliary fluid flow rate, and so on).

Additional information about macro heat exchangers can be found in Overview of the Macro Heat Exchanger Models in the Theory Guide. For the theory behind these models, refer to Macro Heat Exchanger Model Theory in the Theory Guide.

ANSYS Fluent provides two heat transfer models: the default NTU model and the simple effectiveness model. The simple effectiveness model interpolates the effectiveness from the velocity vs. effectiveness curve that you provide. For the NTU model, ANSYS Fluent calculates the effectiveness, ε, from the NTU value that is calculated by ANSYS Fluent from the heat transfer data provided by you in tabular format. ANSYS Fluent will automatically convert this heat transfer data to a primary fluid mass flow rate vs. NTU curve (this curve will be piecewise linear). This curve will be used by ANSYS Fluent to calculate the NTU for macros based on their size and primary fluid flow rate.

The NTU model provides the following features:

- The model can be used to check heat capacity for both the primary and the auxiliary fluid and takes the lesser of the two for the calculation of heat transfer.
- The model can be used to model heat transfer to the primary fluid from the auxiliary fluid, and vice versa.
- The model can be used to model primary fluid-side reverse flow.
- The model can be used with variable density of the primary fluid.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The model can be used in either the serial or parallel ANSYS Fluent solvers.
- The model can be used to make a network of heat exchangers using a heat exchanger group (Using the Grouped Macro Heat Exchanger Model (p. 943)).
- Transient profiles can be used for the auxiliary fluid inlet temperature and for total heat rejection.
- Transient profiles can be used for auxiliary mass flow rates.

The simple effectiveness model provides the following features:

- The model can be used to model heat transfer from the auxiliary fluid to the primary fluid, and vice versa.
- The auxiliary fluid properties can be a function of pressure and temperature, therefore allowing phase change of the auxiliary fluid.
- The model can be used by serial as well as parallel solvers.
- The model can be used to make a network of heat exchangers using a heat exchanger group (Using the Grouped Macro Heat Exchanger Model (p. 943)).
- Transient profiles can be used for the auxiliary fluid inlet temperature and for total heat rejection.
- Transient profiles can be used for auxiliary mass flow rates.

For additional information, see the following sections:

13.3.1. Restrictions
13.3.2. Using the Ungrouped Macro Heat Exchanger Model
13.3.3. Using the Grouped Macro Heat Exchanger Model

13.3.1. Restrictions

The following restrictions exist for the macro heat exchanger models:

- The core must be a 3D mesh with a cross-section that is approximately rectangular in shape.
- The primary fluid streamwise direction (see Equation 6.1 in the Theory Guide) must be aligned with one of the three orthogonal axes defined by the rectangular core.
- The pass-to-pass plane must be perpendicular to the primary fluid streamwise direction.
- The two dimensions of the pass-to-pass plane can each be discretized into multiple macroscopic cells (macros), but in the direction perpendicular to this plane the macros cannot be subdivided.
- It is highly recommended that the free-form **Tet** mesh is not used in the macro heat exchanger model. Instead, evenly distributed **Hex/Wedge** cells should be used for improved accuracy and a more robust solution process.
- Flow acceleration effects are neglected in calculating the pressure loss coefficient.
- For the simple effectiveness model, the primary fluid must have a capacity rate that is less than that of the auxiliary fluid.
- Auxiliary fluid phase change cannot be modeled using the NTU model.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The macro-based method requires that an equal number of cells reside in each macro of equal size and shape.
- The auxiliary fluid flow is assumed to be 1D.
- The pass width has to be uniform.
- Accuracy is not guaranteed when the mesh is not structured or layered.
- Accuracy is not guaranteed when there is upstream diffusion of temperature at the inlet/outlet of the core.
- Non-conformal meshes cannot be attached to the inlet/outlet of the core. An extra layer has to be created to avoid it.

13.3.2. Using the Ungrouped Macro Heat Exchanger Model

The heat exchanger model settings may be written into and read from the boundary conditions file (Reading and Writing Boundary Conditions (p. 56)) using the text commands, `file/write-settings` and `file/read-settings`, respectively. Otherwise, the steps for setting up the ungrouped macro heat exchanger model is as follows:

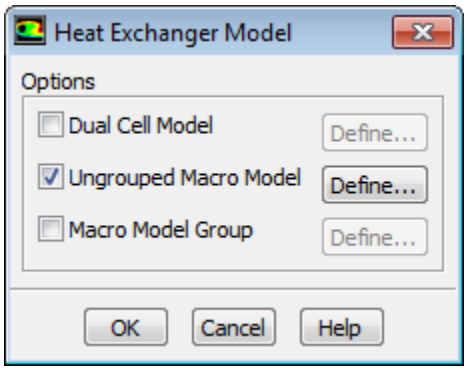
1. Read the mesh and make sure that the inlet and outlet for the heat exchanger are two separate zones.
2. Enable the calculation of energy in the Energy Dialog Box (p. 2175).

   Setup → Models → Energy **ON**

3. Enable the **Ungrouped Macro Model** option and click the **Define...** button in the Heat Exchanger Model Dialog Box (p. 2200) (Figure 13.12: The Heat Exchanger Model Dialog Box (p. 932)) to access the **Ungrouped Macro Heat Exchanger** dialog box.

   Setup → Models → Heat Exchanger **Edit...**

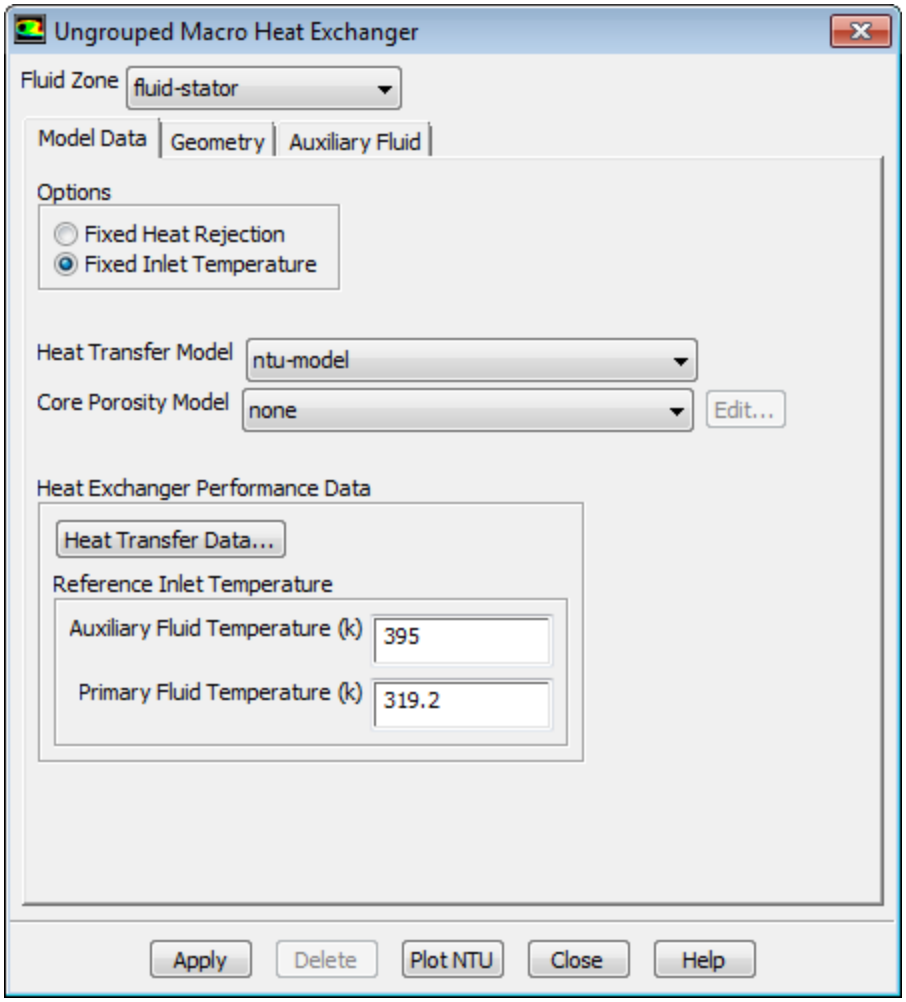
   Figure 13.12: The Heat Exchanger Model Dialog Box



4. Specify the heat exchanger inputs in the **Ungrouped Macro Heat Exchanger** dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.13: The Ungrouped Macro Heat Exchanger Dialog Box Displaying the Model Data Tab



a. In the **Fluid Zone** drop-down list, select the fluid zone representing the heat exchanger core.

b. Under the **Model Data** tab, choose **Fixed Heat Rejection** or **Fixed Inlet Temperature**, as required (Figure 13.13: The Ungrouped Macro Heat Exchanger Dialog Box Displaying the Model Data Tab (p. 933)).

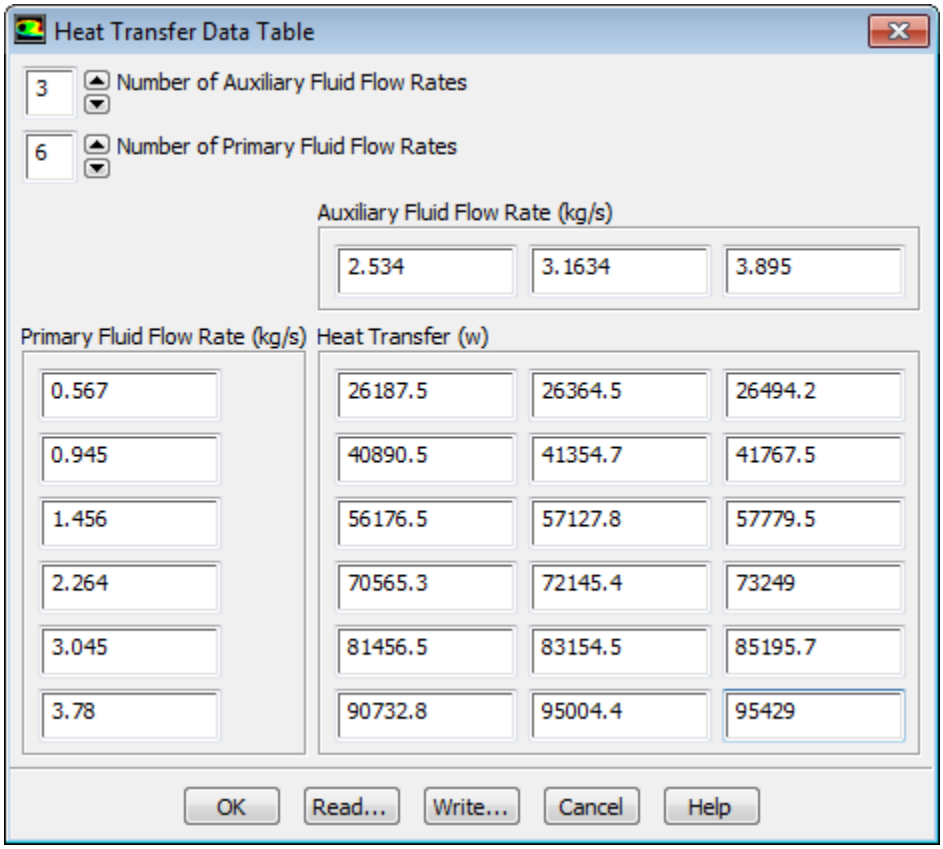
c. Specify the **Heat Transfer Model** as either the default **ntu-model** or the **simple-effectiveness-model**. See step 3 in Choosing a Heat Exchanger Model (p. 920) for details about the differences between these two models.

d. Specify the **Core Porosity Model** if you want ANSYS Fluent to use the pressure loss coefficient function to automatically compute (and update) the porous media coefficients in the cell zones condition dialog box, as described in Streamwise Pressure Drop in the Theory Guide. More information is available in Setting the Pressure-Drop Parameters and Effectiveness (p. 941).

e. If the **ntu-model** is chosen, a **Heat Transfer Data...** button will appear under **Heat Exchanger Performance Data**. Clicking the **Heat Transfer Data...** button will open the Heat Transfer Data Table Dialog Box (p. 2204), where you will enter experimental data that defines how heat transfer values relate to the fluid flow rates (Figure 13.8: The Heat Transfer Data Table Dialog Box (p. 927)). See Specifying Heat Exchanger Performance Data (p. 937) more details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.14: The Heat Transfer Data Table Dialog Box for the NTU Model



f. Enter the **Auxiliary Fluid Temperature** and the **Primary Fluid Temperature** for the **ntu-model**. These are the fixed inlet temperatures at which the test was performed to obtain the heat transfer data.

g. If the **simple-effectiveness-model** is chosen, then clicking the **Velocity Effectiveness Curve...** button, under the **Heat Exchanger Performance Data**, allows you to set the velocity and corresponding effectiveness for each point. More information is available in Specifying Heat Exchanger Performance Data (p. 937).

h. In the **Geometry** tab, define the heat exchanger macros using the **Number of Passes**, the **Number of Rows/Pass**, and the **Number of Columns/Pass** fields. The **Number of Rows/Pass** is along the auxiliary flow direction (height) and the **Number of Columns/Pass** is defined in the pass-to-pass (width) direction. Also, enter the **Auxiliary Fluid Inlet Direction** and **Pass-to-Pass Direction**. You may want to snap the plane tool to either the inlet or outlet of the heat exchanger using the **Update from Plane Tool**. Note that the plane tool must be attached exactly and oriented so that its green arrow points in the auxiliary flow direction, and its blue arrow points in the pass-to-pass direction.

Important

To attach the plane tool exactly, exact coordinates (printed in the console by probing) of the three corner nodes must be entered in the plane tool (x0, x1, x2) in a specific order. Also, note that x0 to x1 is the auxiliary flow direction and x1 to x2 is the pass-to-pass direction.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

More information is available in Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions (p. 938) and Defining the Macros (p. 938).

Figure 13.15: The Ungrouped Macro Heat Exchanger Dialog Box Displaying the Geometry Tab

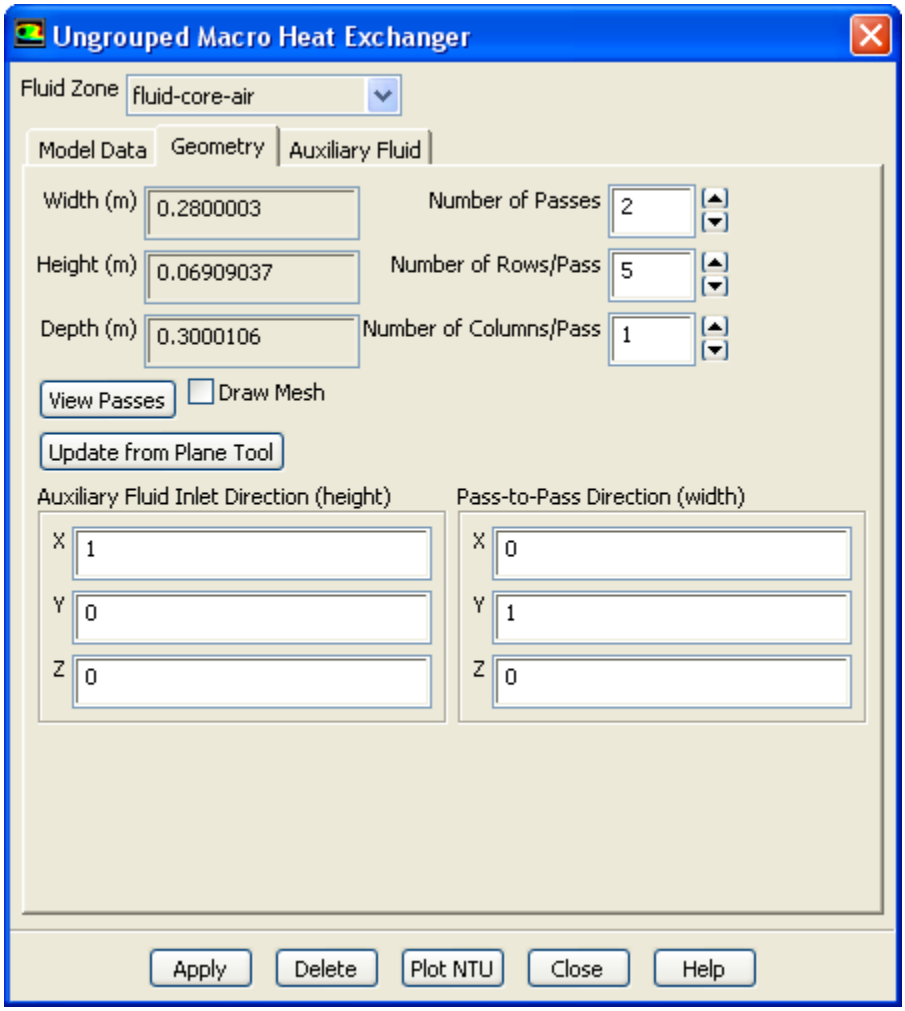
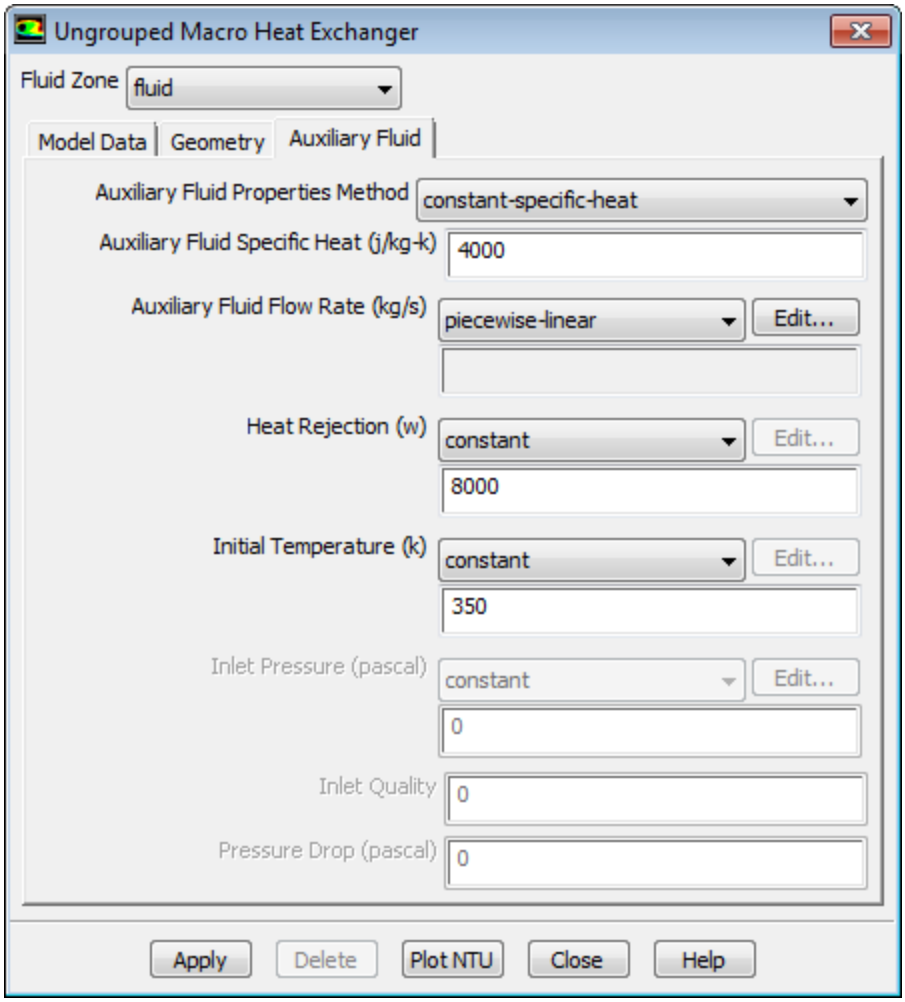


Figure 13.16: The Ungrouped Macro Heat Exchanger Dialog Box Displaying the Auxiliary Fluid Tab



i. In the **Auxiliary Fluid** tab, specify the **Auxiliary Fluid Properties Method**, either as a **constant-specific-heat** or as a **user-defined-enthalpy**.

j. **Auxiliary Fluid Flow Rate**, **Heat Rejection**, **Inlet Temperature**, and **Inlet Pressure** can be provided as a **constant**, **polynomial** or **piecewise-linear** profile that is a function of time. If **user-defined-enthalpy** is selected as the **Auxiliary Fluid Properties Method**, you will need to specify the **Inlet Quality** and the **Pressure Drop**. More information is available in Specifying the Auxiliary Fluid Properties and Conditions (p. 940).

k. Click **Apply** in the **Ungrouped Macro Heat Exchanger** dialog box to save all the settings. Once you click the **Apply** button, the NTU matrix will be computed from the raw data. Therefore, make sure you click **Apply** at the very end of your setup.

Important

When you click **Apply**, look for any error or warning message in the ANSYS Fluent console. Some of the common errors you may see displayed are due to the NTU computations not converging. In such cases, check that

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- you have entered the data correctly
- the values of the data are reasonable
- the operating condition for the auxiliary fluid flow rate is not too far from the range of the heat transfer data

Other error messages you may encounter may be due to macros not getting any cells assigned to it. In such cases, make sure that

- the heat exchanger core is rectangular
- the directions are correct
- you are using uniformly spaced cells in both directions
- the mesh is either a hexahedra or wedge and is structured

l. Repeat steps (a)–(k) for any other heat exchanger fluid zones.

To use multiple fluid zones to define a single heat exchanger, or to connect the auxiliary fluid flow path among multiple heat exchangers, see Using the Grouped Macro Heat Exchanger Model (p. 943).

For additional information, see the following sections:
13.3.2.1. Selecting the Zone for the Heat Exchanger
13.3.2.2. Specifying Heat Exchanger Performance Data
13.3.2.3. Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions
13.3.2.4. Defining the Macros
13.3.2.5. Specifying the Auxiliary Fluid Properties and Conditions
13.3.2.6. Setting the Pressure-Drop Parameters and Effectiveness

13.3.2.1. Selecting the Zone for the Heat Exchanger

Choose the fluid zone for which you want to define a heat exchanger in the **Fluid Zone** drop-down list.

13.3.2.2. Specifying Heat Exchanger Performance Data

Based on the heat transfer model you choose in the **Model Data** tab, some performance data must be entered for the heat exchanger.

- **ntu-model**: For the **ntu-model** you will provide the heat transfer for different primary and auxiliary fluid flow rates. Click the **Heat Transfer Data...** button to open up a tabular dialog box. Set the number of auxiliary flow rates and primary fluid flow rates. The dialog box will resize itself accordingly. You will need to provide various primary fluid flow rates and auxiliary fluid flow rates and the corresponding heat transfer values. You may write this data to a file that can be read later.
- **simple-effectiveness-model**: For this model, you will need to provide velocity versus effectiveness data. To provide this you can click the **Velocity Effectiveness Curve...** button. This will open up a tabular dialog box. In this dialog box, you can set the number of points in the curve, then you can provide velocities and corresponding effectiveness values (note that the effectiveness values must be within the range of 0–1). This data can be written to a file and read back.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.3.2.3. Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions

To define the auxiliary fluid direction and flow path, you will specify direction vectors for the **Auxiliary Fluid Inlet Direction** and the **Pass-to-Pass Direction** in the **Geometry** tab. Figure 13.17: 1x4x3 Macros (p. 939) shows these directions relative to the macros.

For some problems in which the principal axes of the heat exchanger core are not aligned with the coordinate axes of the domain, you may not know the auxiliary fluid inlet and pass-to-pass direction vectors a priori. In such cases, you can use the plane tool as follows to help you to determine these direction vectors.

1. "Snap" the plane tool onto the boundary of the heat exchanger core. (Follow the instructions in Initializing the Plane Tool (p. 1725) for initializing the tool to a position on an existing surface.)

2. Translate and rotate the axes of the tool appropriately until they are aligned with the principal directions of the heat exchanger core. The depth direction is determined by the red axis, the height direction by the green axis, and the width direction by the blue axis.

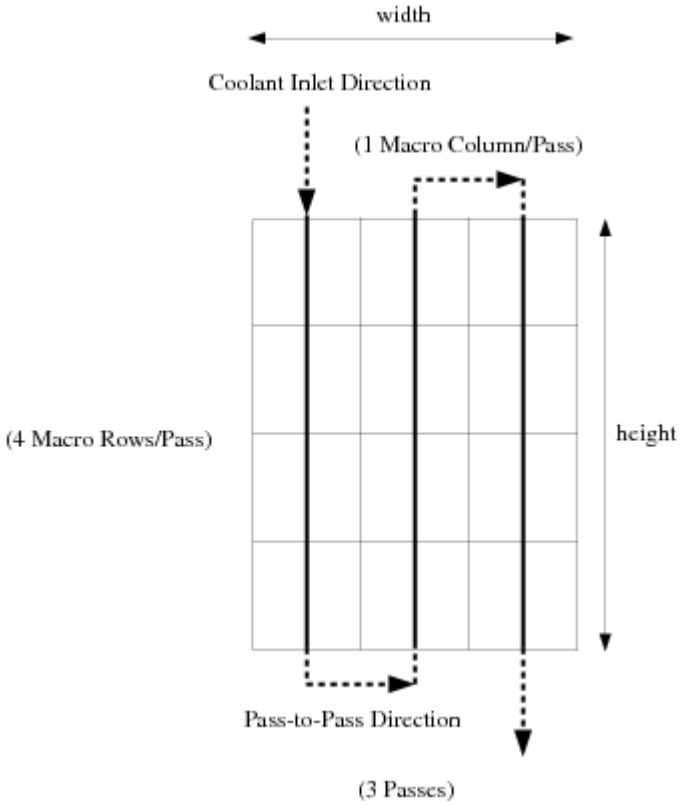
3. Once the axes are aligned, click the **Update from Plane Tool** button in the **Ungrouped Macro Heat Exchanger** dialog box. The directional vectors will be set automatically. (Note that the **Update from Plane Tool** button will also set the height, width, and depth of the heat exchanger core.)

13.3.2.4. Defining the Macros

As discussed in The Macro Heat Exchanger Models (p. 930), the fluid zone representing the heat exchanger core is split into macros. Macros are constructed based on the specified number of passes, the number of macro rows per pass, the number of macro columns per pass, and the corresponding auxiliary fluid inlet and pass-to-pass directions (see Figure 13.17: 1x4x3 Macros (p. 939)). Macros are numbered from 0 to $(n-1)$ in the direction of auxiliary fluid flow, where n is the number of macros.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.17: 1x4x3 Macros



In the **Ungrouped Macro Heat Exchanger** dialog box, in the **Geometry** tab, specify the **Number of Passes**, the **Number of Rows/Pass**, and the **Number of Columns/Pass**. The model will automatically extrude the macros to the depth of the heat exchanger core. For each pass, the **Number of Rows/Pass** are defined in the direction of the auxiliary flow inlet direction and the **Number of Columns/Pass** are defined in the direction of the pass-to-pass direction.

Important

The **Number of Rows/Pass**, as well as the **Number of Columns/Pass** must be divisible by the number of cells in their respective directions. For example, if you have 50 cells in the auxiliary flow direction, you can use 25 for the **Number of Rows/Pass**, but you should not use 26 or 24. If you have 51 cells in that direction, you can only use 51 for the **Number of Rows/Pass**. The same holds true for the other direction.

13.3.2.4.1. Viewing the Macros

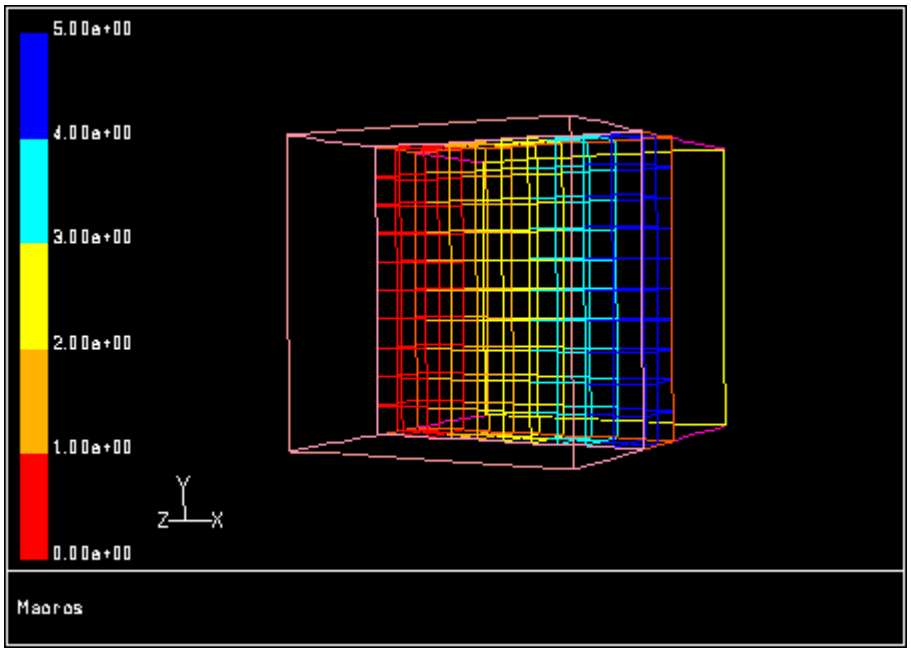
You can view the auxiliary fluid path by displaying the macros. To view the macros for your specified **Number of Passes**, **Number of Rows/Pass**, and **Number of Columns/Pass**, click the **Apply** button at the bottom of the dialog box, then click the **View Passes** button to display it. The path of the auxiliary fluid is color-coded in the display: macro 0 is red and macro $n-1$ is blue.

For some problems, especially complex geometries, you may want to include portions of the computational-domain mesh in your macros plot as spatial reference points. For example, you may want to show the location of an inlet and an outlet along with the macros. This is accomplished by enabling the **Draw Mesh** option. The Mesh Display Dialog Box (p. 2163) will appear automatically when you enable the **Draw Mesh** option, where you can set the mesh display parameters. When you click the **View Passes** button

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in the **Ungrouped Macro Heat Exchanger** dialog box, the mesh display, as defined in the **Mesh Display** dialog box, will be included in the macros plot (see Figure 13.18: Mesh Display With Macros (p. 940)).

Figure 13.18: Mesh Display With Macros



13.3.2.5. Specifying the Auxiliary Fluid Properties and Conditions

To define the auxiliary fluid properties and conditions, you will specify the **Auxiliary Fluid Flow Rate** ($\dot{m}$) in the **Auxiliary Fluid** tab. The properties of the auxiliary fluid can be specified using the **Auxiliary Fluid Properties Method** drop-down list. You can choose a **constant-specific-heat** (c_p) and set the value in the **Auxiliary Fluid Specific Heat** field below, or as a user-defined function for the enthalpy using the **user-defined-enthalpy** option and selecting the corresponding UDF from the **Auxiliary Fluid Enthalpy UDF** drop-down list.

The function should return a single value depending on the index:

- Enthalpy for given values of temperature, pressure, and quality.
- Temperature for given values of enthalpy and pressure
- Specific heat for given values of temperature and pressure

The user-defined function should be of type

```
DEFINE_SOURCE(udf_name, cell_t c, Thread *t, real d[n], int index).
```

where `n` in the expression `d[n]` would be `0` for temperature, `1` for pressure, or `2` for quality. The variable `index` is `0` for enthalpy, `1` for temperature, or `2` for specific heat. This user-defined function should return

```
real value; /* (temperature or enthalpy or Cp depending on index). */
```

- If you want ANSYS Fluent to compute the auxiliary fluid inlet temperature for a specified heat rejection, follow the steps below:

  1. Enable the **Fixed Heat Rejection** option in the **Model Data** tab.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Specify the **Heat Rejection** (q_{total} in Equation 6.15 in the Theory Guide) in the **Auxiliary Fluid** tab.

3. Specify the **Initial Temperature**, which will be used by ANSYS Fluent as an initial guess for the inlet temperature ($T_{in,auxiliary}$ in Equation 6.11 and Equation 6.13 in the Theory Guide).

- If you want ANSYS Fluent to compute the total heat rejection of the core for a given inlet auxiliary fluid temperature, follow the steps below:

  1. Enable the **Fixed Inlet Temperature** option in the **Model Data** tab.

  2. Specify the **Inlet Temperature** ($T_{in,auxiliary}$ in Equation 6.11 and Equation 6.13 in the Theory Guide) in the **Auxiliary Fluid** tab.

- If you enable the **User Defined Enthalpy** option under the **Auxiliary Fluid Properties Method**, you must also specify the **Inlet Pressure** (p_{in} in Equation 6.21 in the Theory Guide) and **Inlet Quality** (x in Equation 6.20 in the Theory Guide).

13.3.2.6. Setting the Pressure-Drop Parameters and Effectiveness

The pressure drop parameters and effectiveness define the **Core Porosity Model**. If you would like ANSYS Fluent to set the porosity of this a heat exchanger zone using a particular core model, you can select the appropriate model. This will automatically set the porous media inputs. There are three ways to specify the **Core Porosity Model** parameters:

- Use the values in ANSYS Fluent's default model.
- Define a new core porosity model with your own values.
- Read a core porosity model from an external file.

If you do not choose a core porosity model, you will need to set the porosity parameters in the cell zone conditions dialog box for the heat exchanger zone(s). To do this, follow the procedures described in User Inputs for Porous Media (p. 267).

The models you define will be saved in the case file.

13.3.2.6.1. Using the Default Core Porosity Model

ANSYS Fluent provides a default model for a typical heat exchanger core. The **default-model** core porosity model is a list of constant values from the **Ungrouped Macro Heat Exchanger** dialog box. These constants are used for setting the porous media parameters. To use these values, simply retain the selection of **default-model** in the **Core Porosity Model** drop-down list in the **Ungrouped Macro Heat Exchanger** dialog box. (You can view the default parameters as described below.)

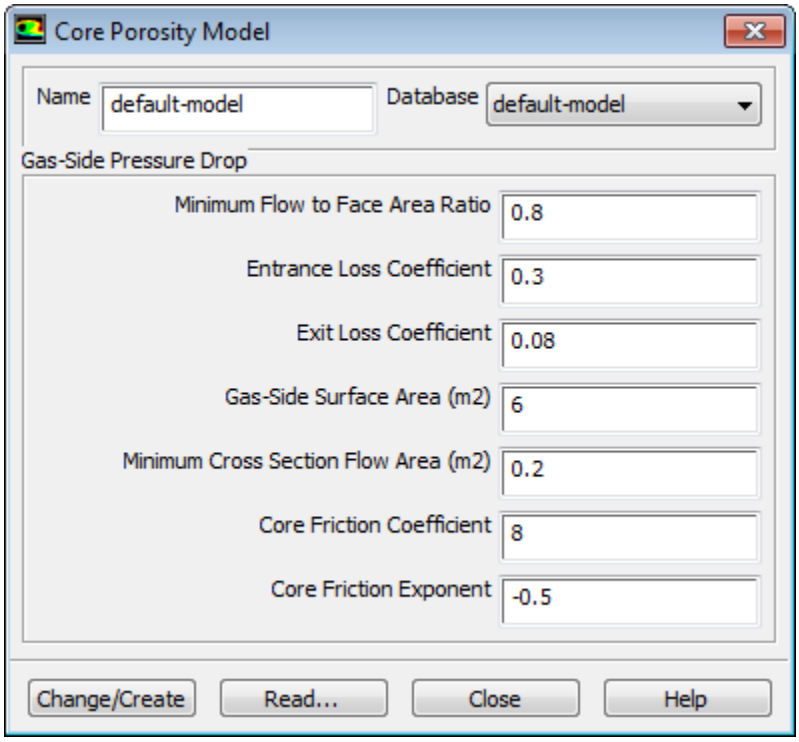
13.3.2.6.2. Defining a New Core Porosity Model

If you want to define pressure-drop and effectiveness parameters that are different from those in the default core porosity model, you can create a new model. The steps for creating a new model are as follows:

1. Click the **Edit...** button to the right of the **Core Porosity Model** drop-down list, for which **default-model** should have been selected. This will open the Core Porosity Model Dialog Box (p. 2211) (Figure 13.19: The Core Porosity Model Dialog Box (p. 942)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.19: The Core Porosity Model Dialog Box



2. Enter the name of your new model in the **Name** box at the top of the dialog box.

3. Under **Gas-Side Pressure Drop**, specify the following parameters used in Equation 6.2 in the Theory Guide:

   - **Minimum Flow to Face Area Ratio** (σ)
   - **Entrance Loss Coefficient** (K_c)
   - **Exit Loss Coefficient** (K_e)
   - **Gas-Side Surface Area** (A)
   - **Minimum Cross Section Flow Area** (A_c)

   and the **Core Friction Coefficient** and **Core Friction Exponent** (a and b, respectively, in Equation 6.3 in the Theory Guide).

4. Click the **Change/Create** button. This will add your new model to the database.

13.3.2.6.3. Reading Heat Exchanger Parameters from an External File

You can read parameters for your **Core Porosity Model** from an external file. A sample file is shown below:

```
("modelname"
 (0.73 0.43 0.053 5.2 0.33 9.1 0.66))
```

The first entry in the file is the name of the model (for example, `modelname`). The second set of numbers contains the gas-side (primary-side) pressure drop parameters:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$(\sigma\ K_c\ K_e\ A\ A_c\ a\ b)$

To read an external heat exchanger file, you will follow these steps:

1. In the **Core Porosity Model** dialog box, click the **Read...** button.

2. In the resulting **Select File** dialog box, specify the **HXC Parameters File** name and click **OK**. ANSYS Fluent will read the core porosity model parameters, and add the new model to the database.

13.3.2.6.4. Viewing the Parameters for an Existing Core Model

To view the parameters associated with a core porosity model that you have already defined, select the model name in the **Database** drop-down list (in the **Core Porosity Model** dialog box). The values for that model from the database will be displayed in the **Core Porosity Model** dialog box.

13.3.3. Using the Grouped Macro Heat Exchanger Model

To define a single heat exchanger that uses multiple fluid zones, or to connect the auxiliary fluid flow path among multiple heat exchangers, you can use heat exchanger groups. To use heat exchanger groups, perform the following steps:

1. Read the mesh and make sure that the inlet and outlet for the heat exchanger are two separate zones.

2. Enable the calculation of energy in the Energy Dialog Box (p. 2175).

   Setup → Models → Energy → ON

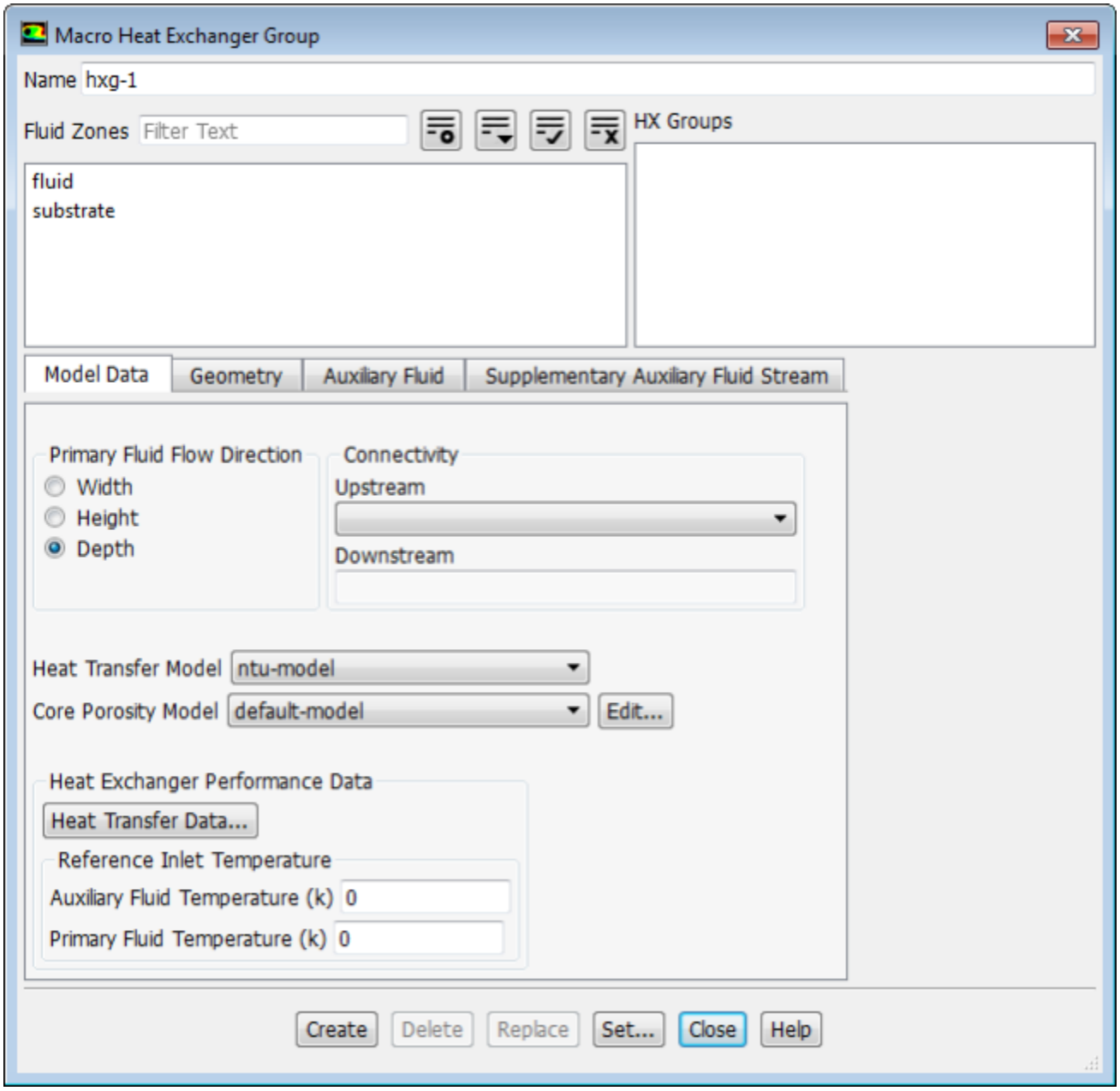
3. Enable the **Macro Model Group** option and click the **Define...** button in the Heat Exchanger Model Dialog Box (p. 2200) to access the **Macro Heat Exchanger Group** dialog box.

   Setup → Models → Heat Exchanger → Edit...

4. Specify the inputs to the heat exchanger group in the **Macro Heat Exchanger Group** dialog box (Figure 13.20: The Macro Heat Exchanger Group Dialog Box (p. 944)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.20: The Macro Heat Exchanger Group Dialog Box



a. Enter the **Name** of the heat exchanger group.

b. Under **Fluid Zones**, select the fluid zones that you want to define in the heat exchanger group (Selecting the Fluid Zones for the Heat Exchanger Group (p. 949)).

c. Click the **Model Data** tab.

   i. Under **Primary Fluid Flow Direction**, specify the primary fluid flow direction as either **Width**, **Height**, or **Depth**.

   ii. Under **Connectivity**, select the **Upstream** heat exchanger group if such a connection exists (see Selecting the Upstream Heat Exchanger Group (p. 949)).

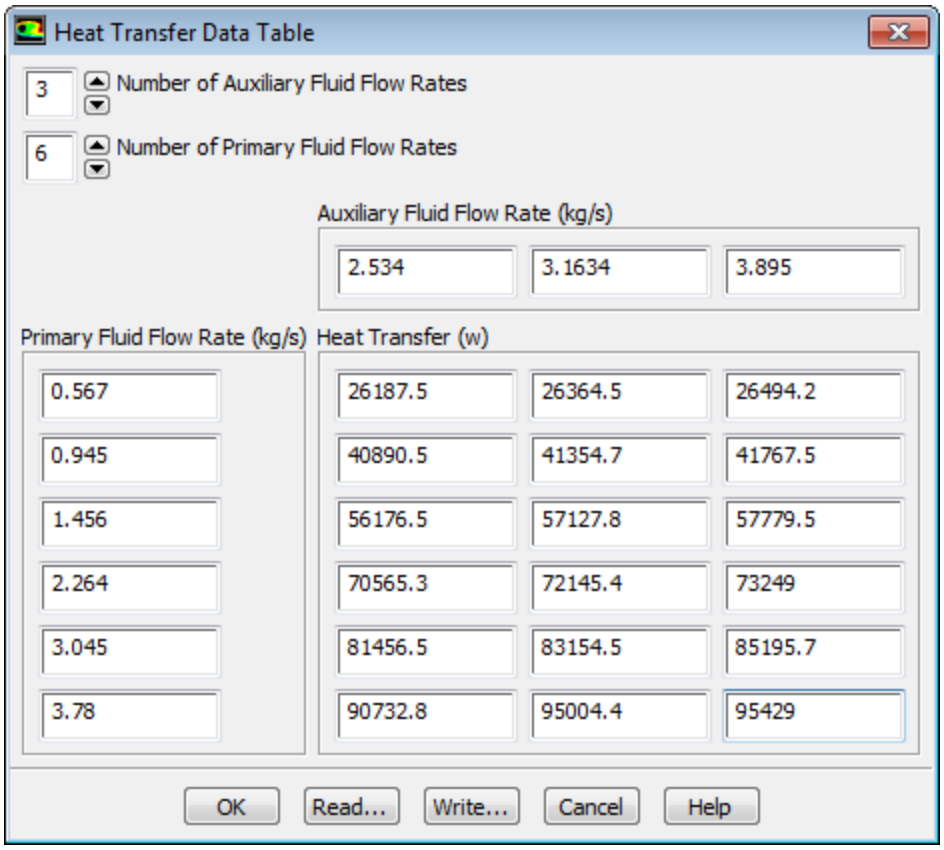
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

iii. In the **Heat Transfer Model** drop-down list, choose either the **ntu-model** or the **simple-effectiveness-model**. See step 3 in Choosing a Heat Exchanger Model (p. 920) for details about the differences between these two models.

iv. From the **Core Porosity Model** drop-down list, specify the core model that should be used to calculate the porous media parameters for the zones in the group. More information is available in Setting the Pressure-Drop Parameters and Effectiveness (p. 941).

v. If the **ntu-model** is chosen, a **Heat Transfer Data...** button will appear under **Heat Exchanger Performance Data**. Clicking the **Heat Transfer Data...** button will open the Heat Transfer Data Table Dialog Box (p. 2204), where you will enter experimental data that defines how heat transfer values relate to the fluid flow rates (Figure 13.8: The Heat Transfer Data Table Dialog Box (p. 927)). See Specifying Heat Exchanger Performance Data (p. 937) more details.

Figure 13.21: The Heat Transfer Data Table Dialog Box for the NTU Model



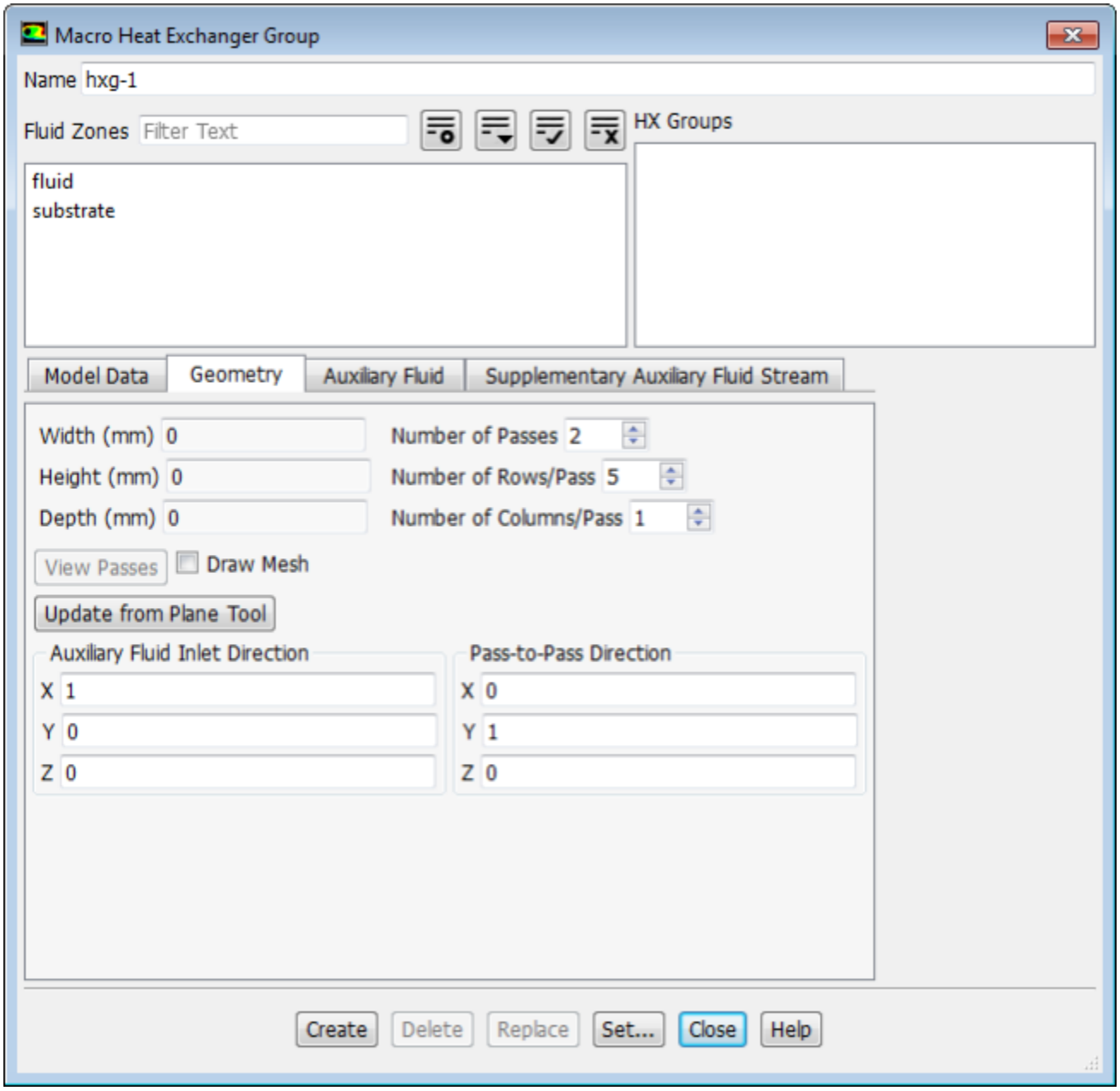
vi. Enter the **Auxiliary Fluid Temperature** and the **Primary Fluid Temperature** for the **ntu-model**. These are the fixed inlet temperatures at which the test was performed to obtain the heat transfer data.

vii. If the **simple-effectiveness-model** is chosen, then clicking the **Velocity Effectiveness Curve...** button, under the **Heat Exchanger Performance Data**, allows you to set the velocity and corresponding effectiveness for each point. More information is available in Specifying Heat Exchanger Performance Data (p. 937).

d. Click the **Geometry** tab (Figure 13.22: The Macro Heat Exchanger Group Dialog Box - Geometry Tab (p. 946)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.22: The Macro Heat Exchanger Group Dialog Box - Geometry Tab



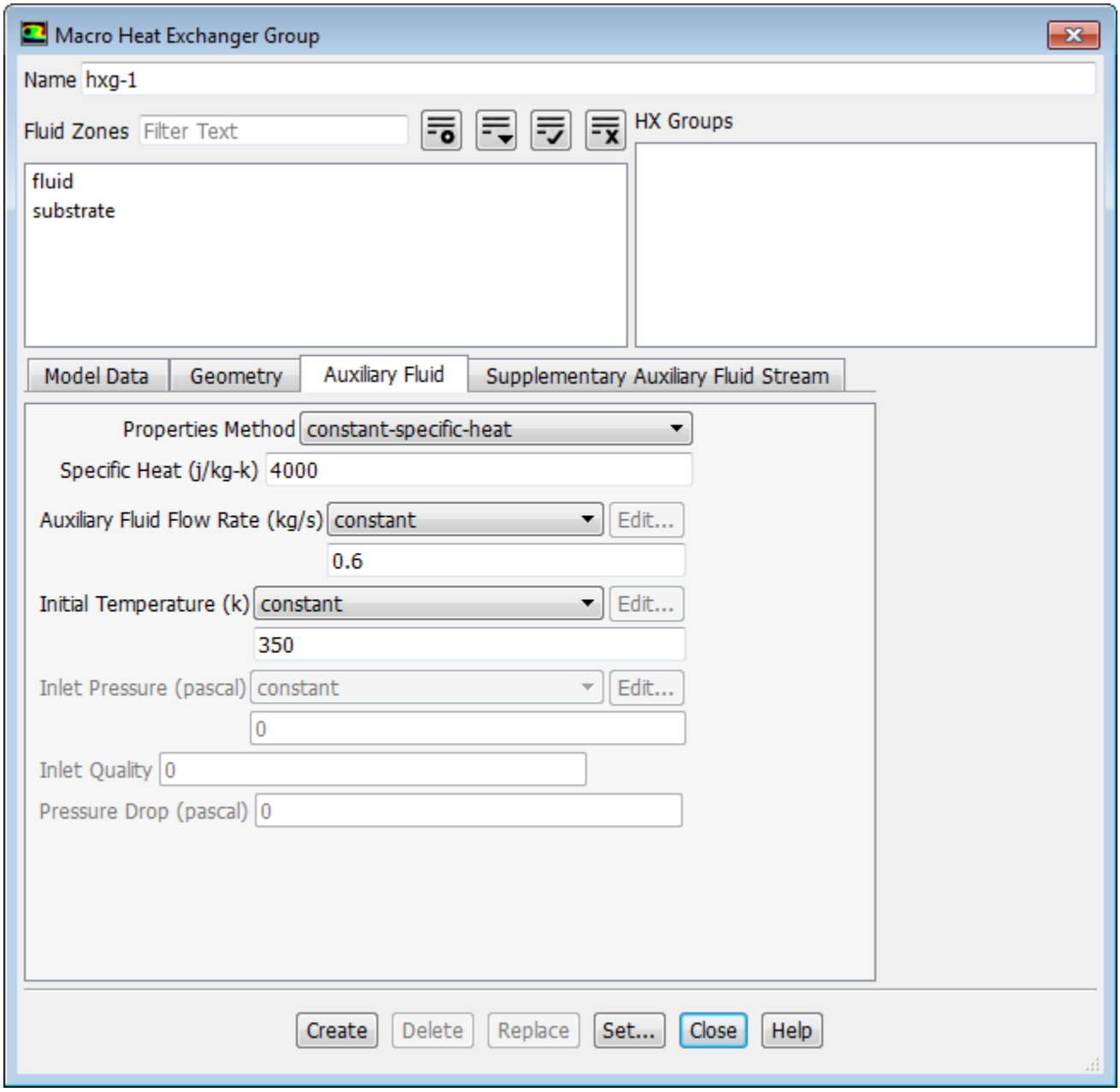
i. Define the heat exchanger macros by specifying the **Number of Passes**, the **Number of Rows/Pass**, and the **Number of Columns/Pass**. More information is available in Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions (p. 938) and Defining the Macros (p. 938).

ii. Specify the **Auxiliary Fluid Inlet Direction** and **Pass-to-Pass Direction** (see Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions (p. 950)).

e. Click the **Auxiliary Fluid** tab (Figure 13.23: The Macro Heat Exchanger Group Dialog Box - Auxiliary Fluid Tab (p. 947)) to specify the auxiliary fluid operating conditions.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.23: The Macro Heat Exchanger Group Dialog Box - Auxiliary Fluid Tab



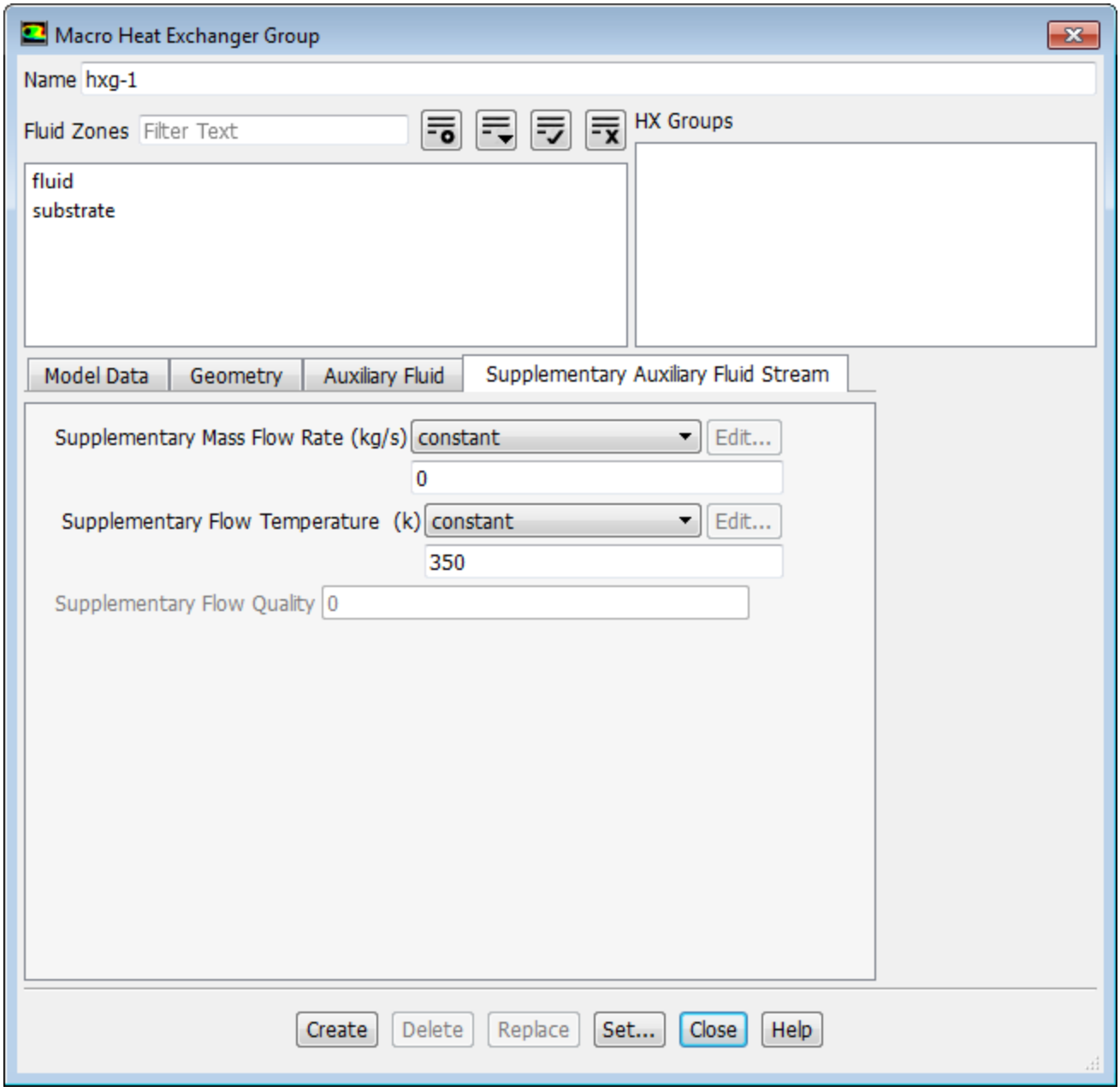
i. Specify the **Specific Heat** as either a **constant-specific-heat** or as a **user-defined-enthalpy**.

ii. **Auxiliary Fluid Flow Rate**, **Initial Temperature**, and **Inlet Pressure** can be provided as a **constant**, **polynomial** or **piecewise-linear** profile that is a function of time (see Specifying the Auxiliary Fluid Properties and Conditions (p. 940)).

f. If a supplementary auxiliary stream is to be modeled, click the **Supplementary Auxiliary Fluid Stream** tab.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.24: The Macro Heat Exchanger Group Dialog Box - Supplementary Auxiliary Fluid Stream Tab



i. You can specify the **Supplementary Mass Flow Rate** as a **constant**, **polynomial** or **piecewise-linear** profile that is a function of time.

ii. You can specify the **Supplementary Flow Temperature** as a **constant**, **polynomial** or **piecewise-linear** profile that is a function of time.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

g. Click **Create** or **Replace** in the **Macro Heat Exchanger Group** dialog box to save all the settings. **Replace** changes the parameters of the already existing group that is selected in the **HX Groups** list.

> **Important**
>
> Creating or replacing any heat exchanger group initializes any previously calculated values for temperature and enthalpy for all macros.

5. If a heat exchanger group comprises multiple fluid zones and you want to override any of the inputs defined in the previous steps, click the **Set...** button to open the **Ungrouped Macro Heat Exchanger** dialog box (Figure 13.13: The Ungrouped Macro Heat Exchanger Dialog Box Displaying the Model Data Tab (p. 933)). Select the particular fluid zone as usual. Notice that the individual heat exchanger inherits the properties of the group by default. You may override any of the following:

   - **Number of Passes**, **Number of Rows/Pass**, and **Number of Columns/Pass**
   - **Auxiliary Fluid Inlet Direction** and the **Pass-to-Pass Direction**
   - **Core Porosity Model**

For additional information, see the following sections:

13.3.3.1. Selecting the Fluid Zones for the Heat Exchanger Group
13.3.3.2. Selecting the Upstream Heat Exchanger Group
13.3.3.3. Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions
13.3.3.4. Specifying the Auxiliary Fluid Properties
13.3.3.5. Specifying Supplementary Auxiliary Fluid Streams
13.3.3.6. Initializing the Auxiliary Fluid Temperature

13.3.3.1. Selecting the Fluid Zones for the Heat Exchanger Group

Select the fluid zones that you want to define in the heat exchanger group in the **Fluid Zones** drop-down list. The auxiliary fluid flow in all these zones will be in parallel. Note that one zone cannot be included in more than one heat exchanger group.

13.3.3.2. Selecting the Upstream Heat Exchanger Group

If you want to connect the current group in series with another group, choose the upstream heat exchanger group. Note that any group can have at most one upstream and one downstream group. Also, a group cannot be connected to itself. Select a heat exchanger group from the **Upstream** drop-down list under **Connectivity** in the **Model Data** tab of the **Macro Heat Exchanger Group** dialog box.

> **Important**
>
> Connecting to an upstream heat exchanger group can be done only while creating a heat exchanger group. The connection will persist even if the connection is later changed and the **Replace** button is clicked. To change a connection to an upstream heat exchanger group, you need to delete the connecting group and create a new heat exchanger group with the proper connection.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.3.3.3. Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions

The **Auxiliary Fluid Inlet Direction** and **Pass-to-Pass Direction**, in the **Geometry** tab can be specified as directed in Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions (p. 938) in the **Ungrouped Macro Heat Exchanger** dialog box. Note that the **Update from Plane Tool** will set the height, width, and depth as the average of the fluid zones selected in the **Fluid Zones**.

13.3.3.4. Specifying the Auxiliary Fluid Properties

The auxiliary fluid can be specified as having a **constant-specific-heat**, or a user-defined function can be written to calculate the enthalpy, as described in Specifying the Auxiliary Fluid Properties and Conditions (p. 940).

13.3.3.5. Specifying Supplementary Auxiliary Fluid Streams

The addition or removal of a supplementary auxiliary fluid is allowed in any of the heat exchanger groups. Note that auxiliary streams are not allowed for individual zones. You will specify the mass flow rate, temperature, and quality of the supplementary auxiliary fluid. You will also need to specify the heat transfer for various flow rates of primary and auxiliary flows. The auxiliary stream has the following assumptions:

- The magnitude of a negative auxiliary stream must be less than the primary auxiliary fluid inlet flow rate of the heat exchanger group.
- Added streams will be assumed to have the same fluid properties as the primary inlet auxiliary fluid.

13.3.3.6. Initializing the Auxiliary Fluid Temperature

When the heat exchanger group is connected to an upstream heat exchanger group, ANSYS Fluent will automatically set the initial guess for the auxiliary fluid inlet temperature, $T_{in,auxiliary}$, to be equal to the $T_{in,auxiliary}$ of the upstream heat exchanger group. Thus the boundary condition $T_{in,auxiliary}$ for the first heat exchanger group in a connected series will automatically propagate as an initial guess for every other heat exchanger group in the series. However, when it is necessary to further improve convergence properties, you will be allowed to override $T_{in,auxiliary}$ for any connected heat exchanger group by providing a value in the **Initial Temperature** field. Whenever such an override is supplied, ANSYS Fluent will automatically propagate the new $T_{in,auxiliary}$ to any heat exchanger groups further downstream in the series. Similarly, every time the $T_{in,auxiliary}$ boundary condition for the first heat exchanger group is modified, ANSYS Fluent will correspondingly update every downstream heat exchanger group.

If you want to impose a non-uniform initialization on the auxiliary fluid temperature field, first connect the heat exchanger groups and then set $T_{in,auxiliary}$ for each heat exchanger group in streamwise order.

All heat exchangers included in a group must use the fixed $T_{in,auxiliary}$ option. All heat exchangers within a heat exchanger group must have the same $T_{in,auxiliary}$. In other words, no local override of this setting is possible through the **Ungrouped Macro Heat Exchanger** dialog box.

13.4. Postprocessing for the Heat Exchanger Model

Postprocessing for the heat exchanger models involves computing the total heat rejection rate by setting up volume monitors and reporting of variables such as computed heat rejection, inlet or outlet temperature, specific heat, and mass flow rate.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that when postprocessing dual cell heat exchanger models using dialog boxes other than the **Heat Exchanger Report** dialog box (which requires you to explicitly specify whether you want reports on either the **Primary** or the **Auxiliary** fluid), attention must be paid to the fact that there are overlapping cell zones in the heat exchanger region. For example:

- When creating an isosurface and making selections from the **From Zones** selection list of the **Iso-Surface** dialog box, you should never select both the primary and the auxiliary fluid zones at the same time. Note that if no selections are made in this list, then all the cell zones are selected.
- When making selections from the **Surfaces** list of the **Contours** dialog box, you should not select a surface that is associated with the primary fluid zone and a surface associated with the auxiliary fluid zone at the same time, if those surfaces are spatially coincident.

For additional information, see the following sections:
13.4.1. Heat Exchanger Reporting
13.4.2. Total Heat Rejection Rate

13.4.1. Heat Exchanger Reporting

Reporting the results for the heat exchanger models is done using the **Heat Exchanger Report** dialog box.

Results → Reports → Heat Exchanger **Edit...**

The following variable options are available for reporting:

- **Computed Heat Rejection**
- **Inlet Temperature**
- **Outlet Temperature**
- **Mass Flow Rate**
- **Specific Heat**

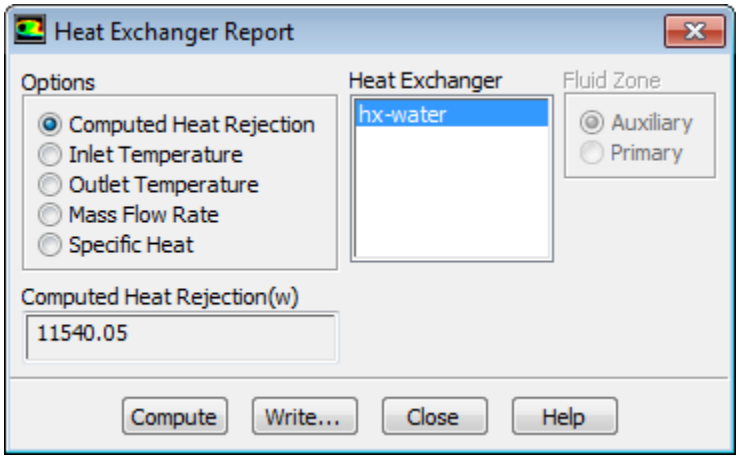
13.4.1.1. Computed Heat Rejection

To display the **Computed Heat Rejection**

1. Select **Computed Heat Rejection** from the **Options** list.
2. Select the **Heat Exchanger** from the selection list.
3. Click **Compute**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.25: The Heat Exchanger Report Dialog Box for Reporting Computed Heat Rejection

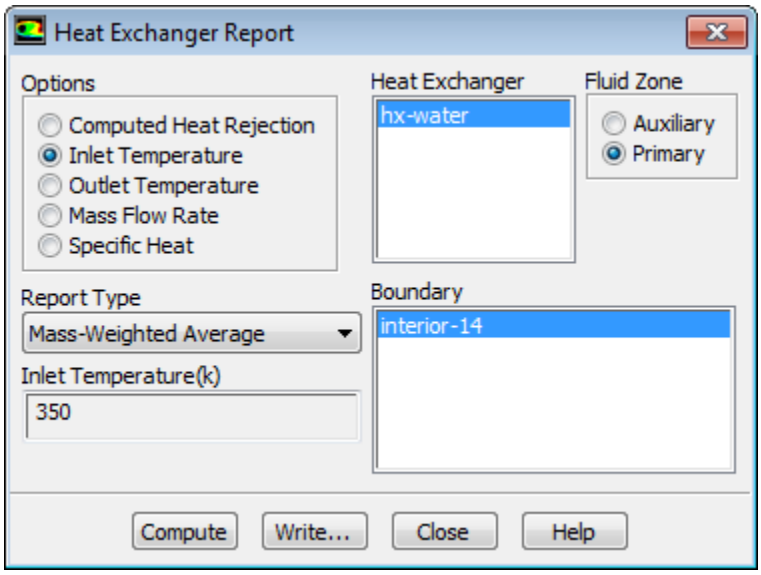


You can write the computed data to a file by clicking the **Write...** button and entering the name of the heat exchanger report file in the **Select File** dialog box.

13.4.1.2. Inlet/Outlet Temperature

Inlet/Outlet Temperature can be reported for both primary and auxiliary fluid in the **Heat Exchanger Report** dialog box.

Figure 13.26: The Heat Exchanger Report Dialog Box for Reporting the Inlet Temperature



1. Select **Inlet Temperature** or **Outlet Temperature** from the **Options** list.
2. Select the heat exchanger from the **Heat Exchanger** selection list.
3. Select either **Auxiliary** or **Primary** as the **Fluid Zone**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Select the appropriate boundary zone and report type from the **Boundary** and **Report Type** lists, respectively.

> **Important**
>
> Note that the macro heat exchangers (unlike the dual cell heat exchanger) do not contain an auxiliary cell zone. Hence, the **Boundary** and **Report Type** fields will not appear in the dialog box if you are reporting an auxiliary inlet/outlet temperature.

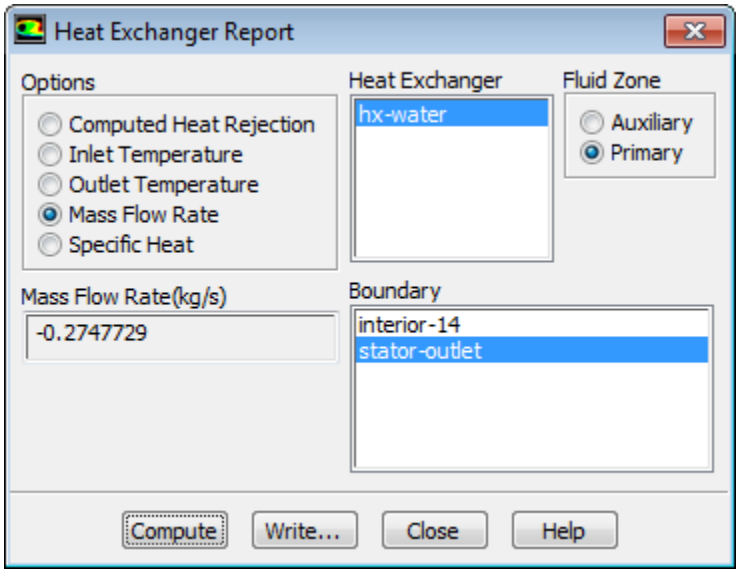
5. Click **Compute**.

You can write the computed data to a file by clicking the **Write...** button and entering the name of the heat exchanger report file in the **Select File** dialog box.

13.4.1.3. Mass Flow Rate

Mass Flow Rate can be reported for both the primary and auxiliary fluid in the **Heat Exchanger Report** dialog box.

Figure 13.27: The Heat Exchanger Report Dialog Box for Reporting Mass Flow Rate



1. Select **Mass Flow Rate** from the **Options** group box.

2. Select the heat exchanger from the **Heat Exchanger** selection list.

3. Select either **Auxiliary** or **Primary** as the **Fluid Zone**.

4. Select the appropriate boundary zone and report type from the **Boundary** and **Report Type** lists, respectively.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click **Compute**.

Important

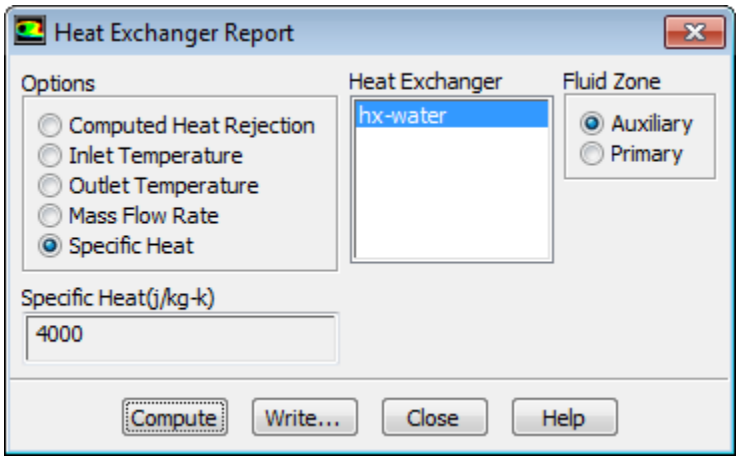
Note that the macro heat exchangers (unlike the dual cell heat exchanger) do not contain an auxiliary cell zone. Hence, the **Boundary** field will not appear in the dialog box if you are reporting an auxiliary fluid mass flow rate.

You can write the computed data to a file by clicking the **Write...** button and entering the name of the heat exchanger report file in the **Select File** dialog box.

13.4.1.4. Specific Heat

Specific Heat for the primary or auxiliary fluid can be reported through the **Heat Exchanger Report** dialog box. If specific heat is defined as a function of temperature, the specific heat reported will be a cell volume averaged value.

Figure 13.28: The Heat Exchanger Report Dialog Box for Reporting Specific Heat



1. Select **Specific Heat** from the **Options** group box.
2. Select the heat exchanger from the **Heat Exchanger** selection list.
3. Select either **Auxiliary** or **Primary** as the **Fluid Zone**.
4. Click **Compute**.

13.4.2. Total Heat Rejection Rate

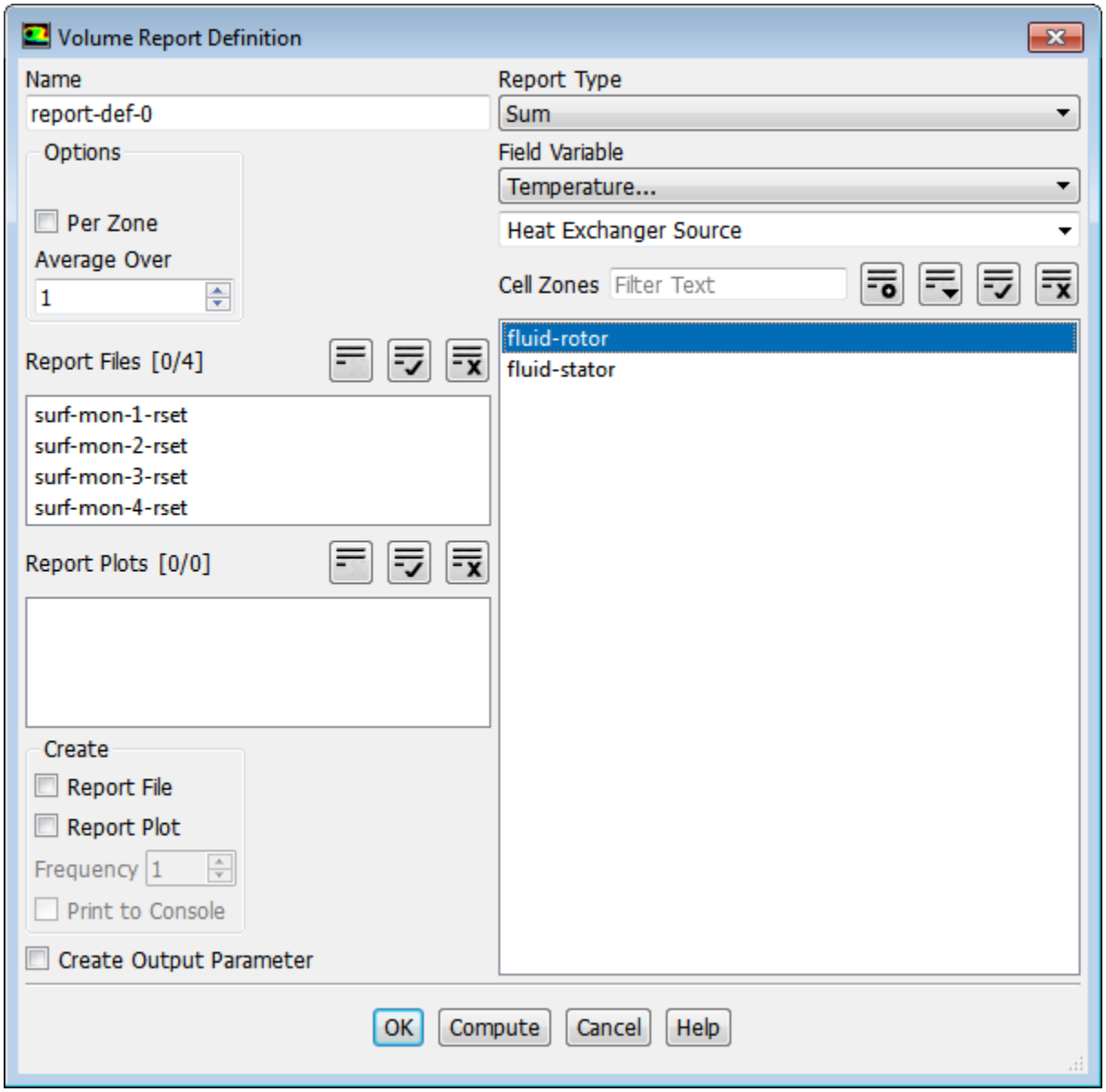
To postprocess the total heat rejection rate, you can set up a volume report definition to monitor convergence and view the computed values.

Solving → Reports → Definitions → New → Volume Report → Sum...

1. Select **Temperature** and **Heat Exchanger Source** from the **Field Variables** drop-down lists.
2. Select the appropriate **Cell Zones** and click **OK** to close the **Volume Monitor** dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.29: The Volume Report Definition Dialog Box



Important

Note that the macro heat exchangers contain only the primary fluid as the cell zone, but in case of the dual cell model, you can select either the primary or auxiliary fluid.

13.5. Useful Reporting TUI Commands

To report the results for the macro heat exchangers, you can use the following text command:

`define` → `models` → `heat-exchanger` → `macro-model` → `heat-exchanger-macro-report`

Specify the fluid zone *id / name* for which you want to obtain information.

To view the connectivity of the heat exchanger groups, use the text command:

```
(report-connectivity)
```

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 14: Modeling Species Transport and Finite-Rate Chemistry

ANSYS Fluent can model the mixing and transport of chemical species by solving conservation equations describing convection, diffusion, and reaction sources for each component species. Multiple simultaneous chemical reactions can be modeled, with reactions occurring in the bulk phase (volumetric reactions) and/or on wall or particle surfaces, and in the porous region. Species transport modeling capabilities, both with and without reactions, and the inputs you provide when using the model are described in this chapter. For theoretical information about species transport, see Species Transport and Finite-Rate Chemistry in the Theory Guide.

Note that you may also want to consider modeling your turbulent reacting flame using the mixture fraction approach (for non-premixed systems, described in Modeling Non-Premixed Combustion (p. 1025)), the reaction progress variable approach (for premixed systems, described in Modeling Premixed Combustion (p. 1089)), the partially premixed approach (described in Modeling Partially Premixed Combustion (p. 1099)), or the composition PDF Transport approach (described in Modeling a Composition PDF Transport Problem (p. 1113)). Modeling multiphase species transport and finite-rate chemistry can be found in Modeling Multiphase Flows (p. 1365).

For simulations using ANSYS Fluent reacting flow models, you can specify mixtures consisting of up to 700 chemical species.

The following models and features may not be used with more than 50 species:

- Density-based solver
- Eulerian PDF Transport model
- Melting/solidification model
- Crevice model
- Thermal diffusivity model
- Surface species
- Site species

Information is divided into the following sections:

14.1. Volumetric Reactions
14.2. Wall Surface Reactions and Chemical Vapor Deposition
14.3. Particle Surface Reactions
14.4. Electrochemical Reactions
14.5. Species Transport Without Reactions
14.6. Reacting Channel Model
14.7. Reactor Network Model

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.1. Volumetric Reactions

Information about using species transport and finite-rate chemistry as related to volumetric reactions is presented in the following subsections. For more information about the theoretical background of volumetric reactions, see Volumetric Reactions in the Theory Guide.

14.1.1. Overview of User Inputs for Modeling Species Transport and Reactions
14.1.2. Enabling Species Transport and Reactions and Choosing the Mixture Material
14.1.3. Importing a Volumetric Kinetic Mechanism in CHEMKIN Format
14.1.4. Defining Properties for the Mixture and Its Constituent Species
14.1.5. Setting up Coal Simulations with the Coal Calculator Dialog Box
14.1.6. Defining Cell Zone and Boundary Conditions for Species
14.1.7. Defining Other Sources of Chemical Species
14.1.8. Solution Procedures for Chemical Mixing and Finite-Rate Chemistry
14.1.9. Postprocessing for Species Calculations

14.1.1. Overview of User Inputs for Modeling Species Transport and Reactions

The basic steps for setting up a problem involving species transport and reactions are listed below.

1. Select species transport and volumetric reactions, and specify the mixture material. See Enabling Species Transport and Reactions and Choosing the Mixture Material (p. 960). The mixture material concept is explained in Mixture Materials (p. 959).

2. If you are also modeling wall or particle surface reactions, select wall surface and/or particle surface reactions. See Wall Surface Reactions and Chemical Vapor Deposition (p. 995) Particle Surface Reactions (p. 1001) for details.

3. Check and/or define the properties of the mixture. See Defining Properties for the Mixture and Its Constituent Species (p. 972). Mixture properties include the following:

   - species in the mixture
   - reactions
   - other physical properties (for example, viscosity, specific heat)

4. Check and/or set the properties of the individual species in the mixture. See Defining Properties for the Mixture and Its Constituent Species (p. 972).

5. Set species cell zone and boundary conditions. See Defining Cell Zone and Boundary Conditions for Species (p. 990).

In many cases, you will not need to modify any physical properties of mixture material because the solver gets species properties, reactions, and so on, from the materials database when you choose the mixture material. Some properties, however, may not be defined in the database. You will be warned when you choose your material if any required properties need to be set, and you can then assign appropriate values for these properties. You may also want to check the database values of other properties to be sure that they are correct for your particular application. For details about modifying an existing mixture material or creating a new one from nothing, see Defining Properties for the Mixture and Its Constituent Species (p. 972). Modifications to the mixture material can include the following:

- Addition or removal of species
- Changing the chemical reactions

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Modifying other material properties for the mixture
- Modifying material properties for the mixture's constituent species

If you are solving a reacting flow, you will usually want to define the mixture's specific heat as a function of composition, and the specific heat of each species as a function of temperature. You may want to do the same for other properties as well. By default, most species specific heats in the database are piecewise-polynomial functions of temperature, but you may choose to specify a different temperature-dependent function if you know of one that is more suitable for your problem.

14.1.1.1. Mixture Materials

The concept of mixture materials has been implemented in ANSYS Fluent to facilitate the setup of species transport and reacting flow. A mixture material may be thought of as a set of species and a list of rules governing their interaction. The mixture material carries with it the following information:

- A list of the constituent species, referred to as "fluid" materials
- A list of mixing laws dictating how mixture properties (density, viscosity, specific heat, and so on) are to be derived from the properties of individual species if composition-dependent properties are desired
- A direct specification of mixture properties if composition-independent properties are desired
- Diffusion coefficients for individual species in the mixture
- Other material properties (for example, absorption and scattering coefficients) that are not associated with individual species
- A set of reactions, including a reaction type (finite-rate, eddy-dissipation, and so on) and stoichiometry and rate constants

The mixture materials are stored in the ANSYS Fluent materials database. Many common mixture materials are included (for example, methane-air, propane-air). Generally, one or two-step reaction mechanisms and many physical properties of the mixture and its constituent species are defined in the database.

The ANSYS Fluent materials database is accessed in one of two ways, depending on the sequence of your workflow:

- through the **Species Model** dialog box before any one of the species models is enabled
- through the **Create/Edit Materials** dialog box after one of the available species models is enabled

When you enable one of the species models, the mixture material that you have selected for your application, its constituent fluid materials, and properties will be loaded into your simulation. If any necessary information about the selected material (or the constituent fluid materials) is missing, the solver will inform you that you need to specify it. In addition, you may choose to modify any of the predefined properties. See Using the **Create/Edit Materials** Dialog Box (p. 461) for information about the sources of ANSYS Fluent database property data.

For example, if you plan to model combustion of a methane-air mixture using global kinetics, you do not need to explicitly specify the species involved in the reaction or the reaction itself. You will simply select **methane-air** as the mixture material to be used, and the relevant species (CH_4, O_2, CO_2, H_2O, and N_2) and reaction data will be loaded into the solver from the database. You can then check the species, reactions, and other properties and define any properties that are missing and/or modify any properties for which you want to use different values or functions. You will generally want to define a composition-

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and temperature-dependent specific heat, and you may want to define additional properties as functions of temperature and/or composition.

The use of mixture materials gives you the flexibility to select and load from the Fluent database one of the many predefined mixtures that is appropriate for your case or, if you want to create your own custom mixture material, a mixture-template (the default) consisting of H_2O, O_2, and N_2. Once the mixture material has been copied to the solver, you can modify it in the Create/Edit Materials Dialog Box (p. 2302), as described in Defining Properties for the Mixture and Its Constituent Species (p. 972).

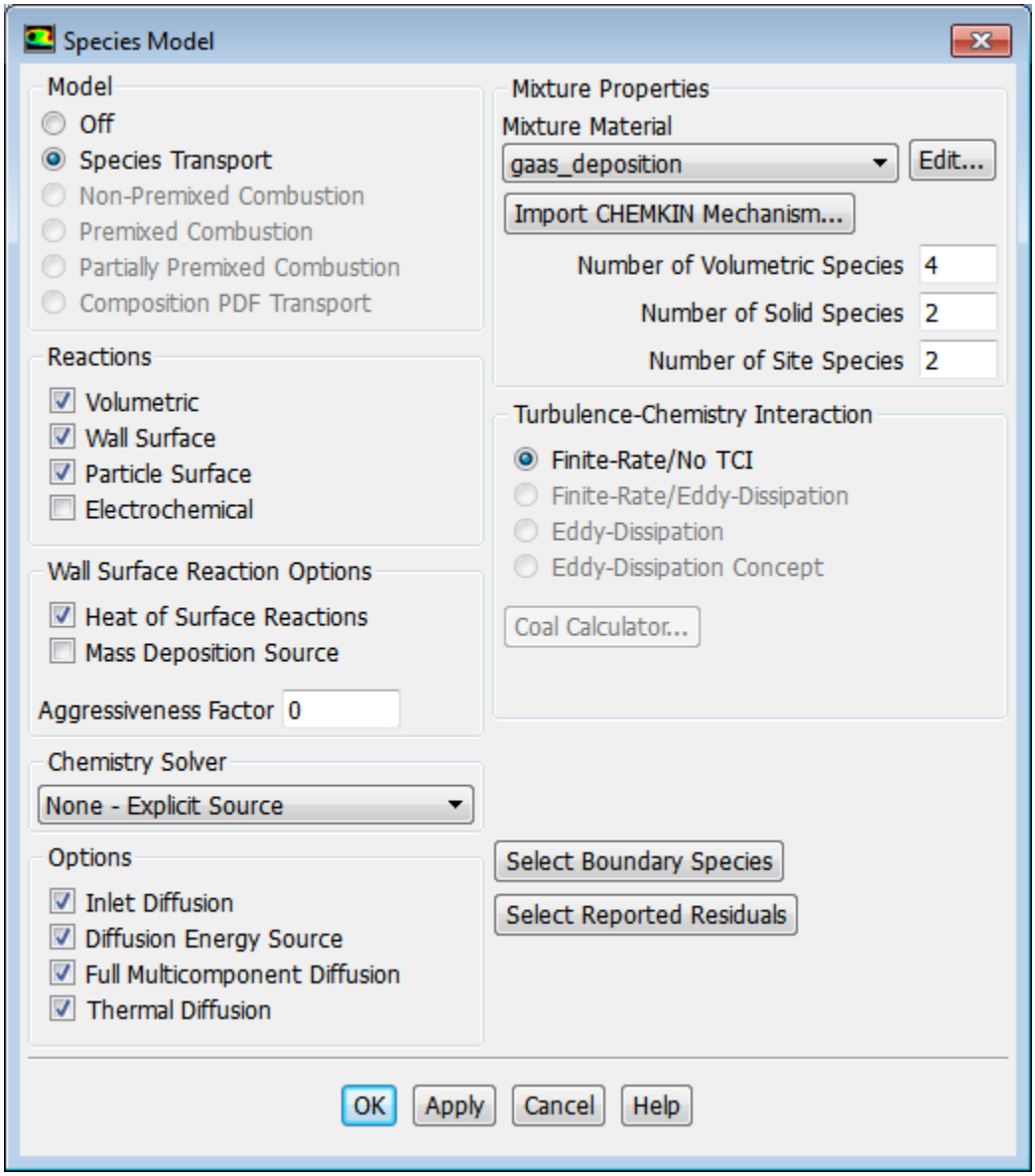
14.1.2. Enabling Species Transport and Reactions and Choosing the Mixture Material

The problem setup for species transport and volumetric reactions begins in the **Species Model** dialog box (Figure 14.1: The Species Model Dialog Box (p. 961)). For cases that involve multiphase species transport and reactions, refer to Modeling Species Transport in Multiphase Flows in the Theory Guide.

Setup → **Models** → **Species** **Edit...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.1: The Species Model Dialog Box



1. Under **Model**, select **Species Transport**.

2. Under **Reactions**, select **Volumetric**.

3. In the **Mixture Material** drop-down list under **Mixture Properties**, select which mixture material you want to use in your problem or, if you want to create your own mixture material, retain the default selection of **mixture-template** consisting of H_2O, O_2, and N_2. The drop-down list includes all of the mixtures that are currently defined in the database. If there is a mixture material listed that is similar to your desired mixture, you may choose that material and modify it following the steps provided in Defining Properties for the Mixture and Its Constituent Species (p. 972).

   The number of species in the selected mixture material is displayed in the **Number of Volumetric Species** field and, if applicable, in the **Number of Solid Species** and **Number of Site Species** fields. To review the mixture composition and reactions of the selected Fluent database mixture material, click the **View...** button to the right of **Mixture Material**. This opens the **Fluent Database Materials** dialog box. Clicking, in turn, the **View...** buttons next to **Mixture Species** and **Reaction** opens the corresponding dialog boxes allowing you to examine the constituent species and reactions data.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note, that the mixture properties cannot be edited before the mixture material is copied to your Fluent problem. For this reason, all related controls are dimmed while viewing the mixture properties.

When you enable the species transport model by clicking either **OK** or **Apply**, ANSYS Fluent performs the following:

- The selected Fluent database mixture material including all constituent fluid materials (species) and properties are automatically copied to your case and appear under **Mixture** in the tree and the **Materials** task page.
- If any properties for the selected mixture material (or the constituent fluid materials) are missing, the solver prints to the console a notice of the required data.
- When the mixture material is copied, the **Mixture Material** drop-down list in the **Species Model** dialog box changes to contain only the mixtures defined for your Fluent problem.
- The **View...** button is replaced by the **Edit...** button giving you an option to access the **Edit Material** dialog box (Edit Material Dialog Box (p. 2355)) from the **Species Model** dialog box.
- The **Create/Edit Materials** dialog box lists **mixture** in the **Material Type** drop-down list allowing you to access the Fluent mixture material database. If you want to check or modify any properties of the mixture material, use the Create/Edit Materials Dialog Box (p. 2302), as described in Defining Properties for the Mixture and Its Constituent Species (p. 972).

You can add more mixture materials to your case by copying them from the database, as described in Copying Materials from the ANSYS Fluent Database (p. 464), or by creating a new mixture, as described in Creating a New Material (p. 466) and Defining Properties for the Mixture and Its Constituent Species (p. 972).

4. Select the **Turbulence-Chemistry Interaction** (TCI) modeling option. The following options are available:

Finite-Rate/No TCI

computes only the Arrhenius rate (see Equation 7.7 in the Theory Guide) and neglects turbulence-chemistry interaction. This model is recommended for laminar or turbulent flows using complex chemistry where either the turbulence time-scales are expected to be fast relative to the chemistry time scales or where the chemistry is sufficiently complex that the chemistry timescales of importance are highly disparate throughout the simulation. You can specify the following inputs:

Flow Iterations per Chemistry Update (steady-state only)

Increasing the number reduces the computational expense of the chemistry calculations. By default, ANSYS Fluent will update the chemistry once per flow iteration. This option is not available when the **None - Explicit Source** option is selected for **Chemistry Solver**.

Aggressiveness Factor (steady-state only)

This is a numerical factor that controls the robustness and the convergence speed. This value ranges between 0 and 1, where 0 is the most robust, but results in the slowest convergence. The default value for the **Aggressiveness Factor** is 0.5. This option is not available when the **None - Explicit Source** option is selected for **Chemistry Solver**.

Finite-Rate/Eddy-Dissipation (turbulent flows only)

computes both the Arrhenius rate and the mixing rate and uses the smaller of the two.

Eddy-Dissipation (turbulent flows only)

computes only the mixing rate (see Equation 7.25 and Equation 7.26 in the Theory Guide).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Eddy-Dissipation Concept (turbulent flows only)
models turbulence-chemistry interaction with detailed chemical mechanisms (see Equation 7.7 and Equation 7.30 in the Theory Guide). When using this model, you can modify the following:

Flow Iterations per Chemistry Update (steady-state only)
Increasing the number reduces the computational expense of the chemistry calculations. By default, ANSYS Fluent will update the chemistry once per flow iteration.

Aggressiveness Factor (steady-state only)
This is a numerical factor that controls the robustness and the convergence speed. This value ranges between 0 and 1, where 0 is the most robust, but results in the slowest convergence. The default value for the **Aggressiveness Factor** is 0.5.

Volume Fraction Constant , Time Scale Constant
C_ξ in Equation 7.28 and C_τ in Equation 7.29 in the Theory Guide. The default values are recommended.

5. From the **Chemistry Solver** drop-down list, select the solver for your simulation (note that the solvers available in the drop-down list are dependant on the selected **Turbulence-Chemistry Interaction** model):

   - If you want the stiff chemistry to relax to chemical equilibrium without solving reactions, select **Relax to Chemical Equilibrium**. This option is not available for **Eddy Dissipation Concept**. For information about the Relaxation to Chemical Equilibrium model, refer to The Relaxation to Chemical Equilibrium Model in the Theory Guide

   Important

   The **Relax to Chemical Equilibrium** chemistry solver requires thermodynamic data of all species to calculate chemical equilibrium. You can supply your own database using the **CHEMKIN Mechanism Import** dialog box (opened by clicking **Import CHEMKIN mechanism...**) (see Importing a Volumetric Kinetic Mechanism in CHEMKIN Format (p. 967) and Importing a Surface Kinetic Mechanism in CHEMKIN Format (p. 995) for details). A sample database is also contained in the file `.../flu-entx.x/cpropep/data/thermo.db`. If you define your own species in ANSYS Fluent, as determined by the **Chemical Formula** in the **Create/Edit Materials** dialog box, and this species name is not in the thermo.db file, ANSYS Fluent will report an error. To overcome this error, you must manually enter all the unknown species in the thermo file.

   - If you want to solve finite-rate chemistry, select either **Stiff Chemistry Solver** or **CHEMKIN-CFD Solver**. These two solvers are available when using the **Finite-Rate/No TCI** or **Eddy-Dissipation Concept** TCI option.

     Selecting **CHEMKIN-CFD Solver** will enable you to use the ANSYS CHEMKIN-CFD solution algorithms, which are based on and compatible with the ANSYS Chemkin-Pro technology. Once you click **OK** or **Apply**, you will be prompted to import a CHEMKIN mechanism, if you have not already done so. Refer to the ANSYS Chemkin-Pro Input and Theory manuals on the ANSYS Customer Portal (support.ansys.com/docdownloads) for details on the chemistry formulation options. See Using ANSYS Encrypted Mechanisms (p. 968) for instructions on establishing a CHEMKIN-mechanism based material. In addition, Appendix D: Controlling CHEMKIN-CFD Solver Parameters Using Text Commands (p. 2875) includes information about advanced solver settings and diagnostic information for the CHEMKIN-CFD solver.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you want to make explicit use of chemistry source terms in the species transport equations, without a stiff-chemistry solver, select **None - Explicit Source**. This default solver is not recommended for stiff or complex chemistry.

6. (optional) For TCI model options that consider finite-rate chemistry, set the integration parameters using the **Integration Parameters** dialog box (opened by clicking the **Integration Parameters...** button). In the **Integration Parameters** dialog box, you can select acceleration methods (see Using Chemistry Acceleration (p. 1127) for more details):

   - **Dynamic Mechanism Reduction** (DMR) (for **Stiff Chemistry Solver**). See Using Dynamic Mechanism Reduction (p. 1131) for details.
   - **Chemistry Agglomeration** (CA) (for **Stiff Chemistry Solver** and **Relax to Chemical Equilibrium**). See Using Chemistry Agglomeration (p. 1136) for details.
   - **ISAT**. See Using ISAT (p. 1128) for details.
   - **Dynamic Cell Clustering** (DCC) (for the **CHEMKIN-CFD Solver** in transient cases). DCC is enabled by default. See see Using Dynamic Cell Clustering (p. 1137) for details.

7. (optional) Select any **Options** desired. If you want to model full multicomponent (Stefan-Maxwell) diffusion or thermal (Soret) diffusion, select the **Full Multicomponent Diffusion** or **Thermal Diffusion** option, respectively.

   See Full Multicomponent Diffusion (p. 520) for details.

8. Enable the **Thickened Flame Model** when modeling unsteady laminar flames, or when using an LES model for turbulent premixed and partially-premixed flames. Specify the **Number of Grid Points in Flame**, which by default are 8 grid points. Consequently, in the **Create/Edit Materials** dialog box, you will specify the **Laminar Flame Speed**, for which you have a choice of **constant**, **user-defined**, or **metghalchi-keck-law**, and the **Laminar Flame Thickness**, for which you have a choice of **constant**, **user-defined**, or **diffusivity-over-flame-speed**.

   Note

   Since the thickened flame model must capture the correct flame speed, it is imperative that the combination of your chemical mechanism and your molecular transport properties (diffusion coefficients and thermal conductivity) reproduce the correct laminar flame speed. It is recommended that you verify this with a 1D premixed flame simulation using ANSYS Chemkin-Pro, for example.

   When the **Thickened Flame Model** is enabled (Figure 14.2: The Species Model Dialog Box Displaying the Thickened Flame Model (p. 965)), you have the option to hook a UDF in the **User-Defined Function Hooks** dialog box to customize the **Thickened Flame Model** parameters. Refer to `DEFINE_THICKENED_FLAME_MODEL` in the *Fluent Customization Manual* for details.

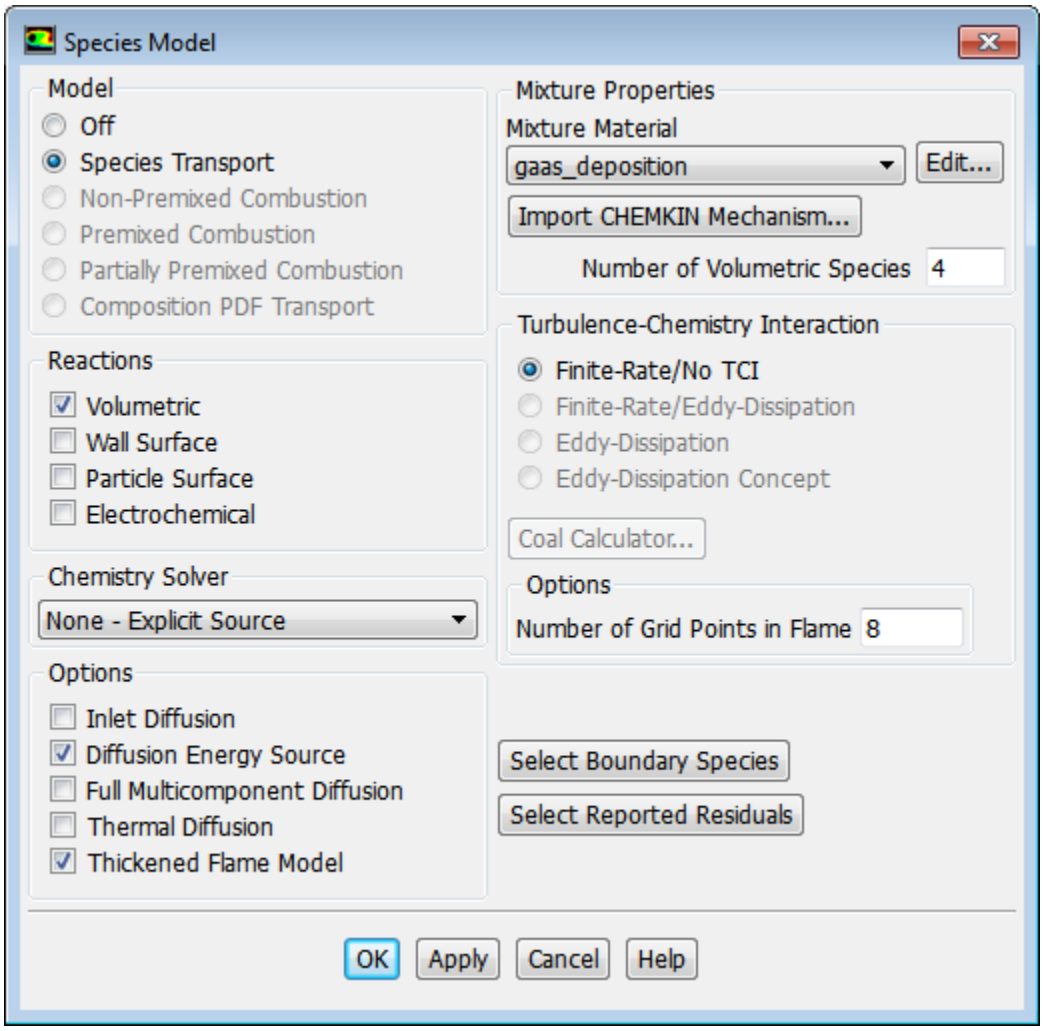
   Important

   The **Thickened Flame Model** is available only for unsteady laminar or turbulent (LES / DES / SAS / SBES / SDES) flows, with **Species Transport** and **Volumetric Reactions** enabled.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For information about the theory behind the **Thickened Flame Model**, refer to The Thickened Flame Model in the Theory Guide.

Figure 14.2: The Species Model Dialog Box Displaying the Thickened Flame Model



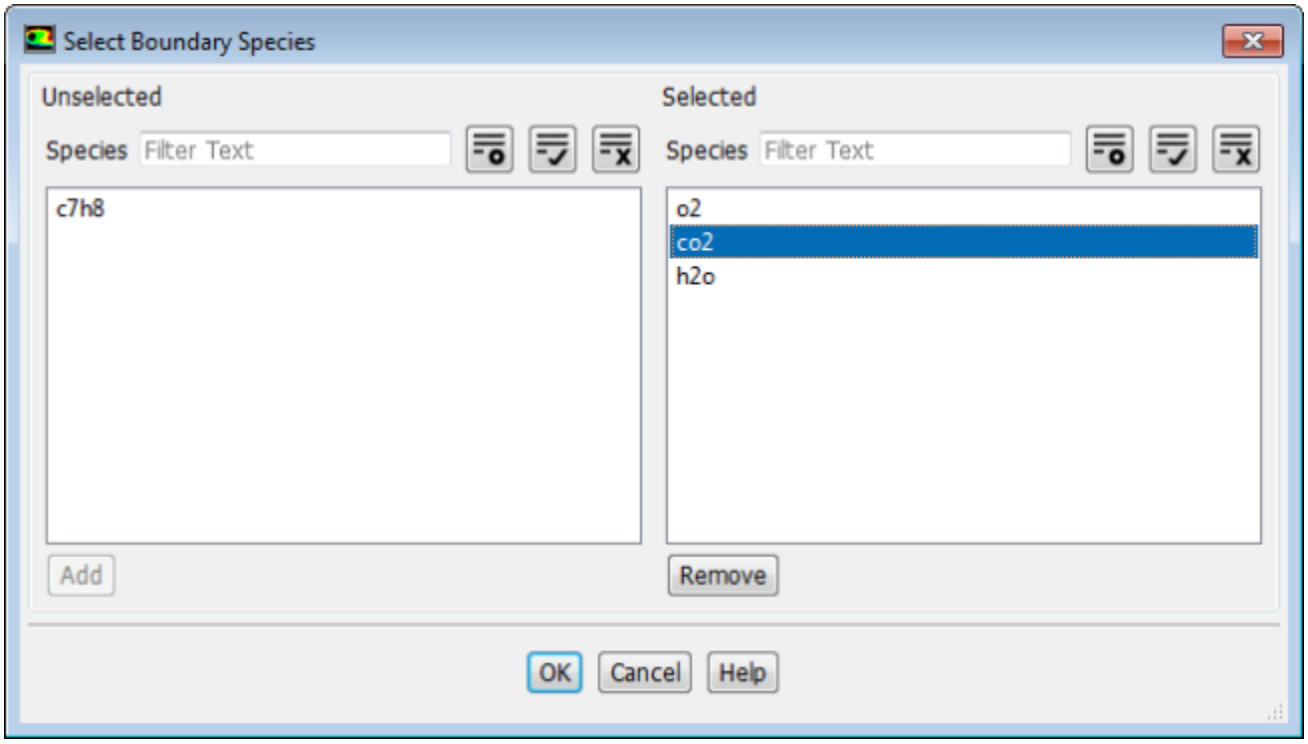
The Species Model Dialog Box

9. (optional) Select the species for which you want to specify boundary conditions. Only the species that you have selected will be listed in the relevant boundary condition dialog boxes. This will facilitate the problem setup, especially when a large number of species are used in a gas mixture. Note that the **Patch** and **Hybrid Initialization** dialog boxes, as well as the **Solution Initialization** task page will also display only the selected species.

   a. Click **Select Boundary Species**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.3: The Select Boundary Species Dialog Box



By default, all the gas species for which the transport equations are to be applied are set as boundary species and appear in the **Selected** multiple-selection list of the **Select Boundary Species** dialog box (see Figure 14.3: The Select Boundary Species Dialog Box (p. 966)).

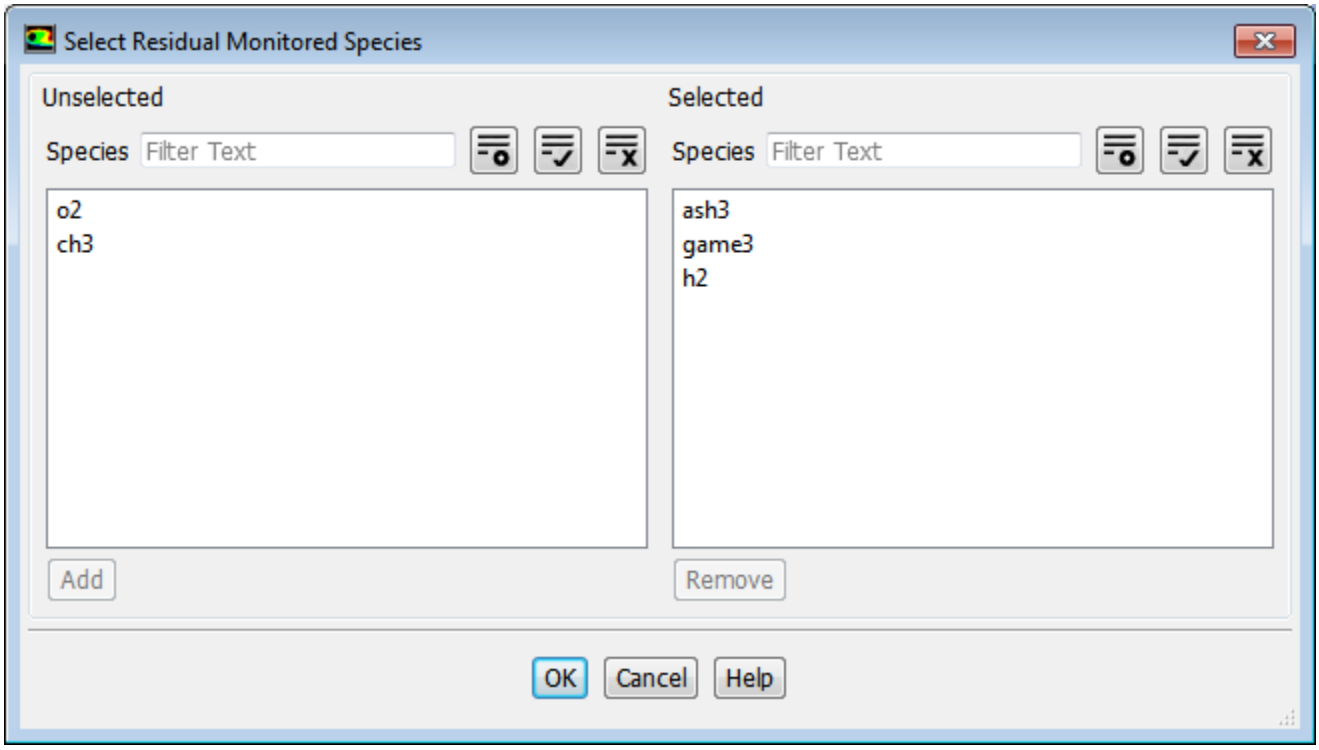
b. Modify the boundary species list as follows:

- To remove the species from the **Selected Species** multiple-selection list, select it and click **Remove**. The species will be moved from the **Selected Species** list to the **Unselected Species** list.
- To add the species back to the **Selected Species** list, simply select it in the **Unselected Species** multiple-selection list and click **Add**. The species will be moved from the **Unselected Species** list to the **Selected Species** list.

10. (optional) In a similar manner, click the **Select Reported Residuals** button and select the species that you want to monitor. Only the residuals for the selected species will be monitored and plotted in the graphics window during the simulation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.4: The Select Residual Monitored Species



By default, all the species for which the species transport equations are to be solved are set as monitored species, and their residual plots will be drawn in the graphics window during the run. For a large number of monitored species, it may be difficult to distinguish the overlapping plot lines. By decreasing the number of monitored species, you can obtain clearer residual plots.

Important

- The Fluent solver performs the convergence checks only for the monitored species. To prevent false convergence, make sure to include the species that may have difficulties converging in the monitored species list.
- You can modify the list of the monitored species dynamically during a run.

14.1.3. Importing a Volumetric Kinetic Mechanism in CHEMKIN Format

You can import gas kinetics mechanism in CHEMKIN format, which can be used with any solver option. For the **CHEMKIN-CFD Solver** and **Stiff Chemistry Solver**, native CHEMKIN computations will determine reaction rates, so full compatibility with ANSYS Chemkin-Pro is provided.

Note

If a mechanism includes at least one reaction type that is not supported in the Fluent material-definition interface (that is, it is not included in the options for manual setup of reactions), the reactions of the CHEMKIN mechanism will not be displayed in the Fluent interface, even though they will be correctly used in Fluent computations.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.1.3.1. Using ANSYS Encrypted Mechanisms

For certain types of simulations, Fluent supports the use of encrypted gas kinetics mechanisms, such as those derived from the ANSYS Model Fuels Library (encrypted) or through ANSYS services. In order to use an encrypted mechanism, you must first import both the gas-phase kinetics and thermodynamic files as described in Procedure for Importing Volumetric CHEMKIN Mechanisms (p. 968).

When using encrypted CHEMKIN gas mechanisms, note the following points:

- Encrypted mechanisms can only be used with the CHEMKIN-CFD solver, the stiff chemistry solver, or the **Relax to Chemical Equilibrium** option
- Prior to importing a flamelet or reading a PDF table generated with an encrypted mechanism, you must first import the mechanism
- Chemical equilibrium PDF table generation is not supported
- Multiphase and Decoupled Detailed Chemistry models are not supported
- You cannot combine encrypted and plain-text gas kinetics, surface kinetics and thermodynamics files in a simulation
- The chemistry acceleration options Dynamic Mechanism Reduction (DMR) and Dimension Reduction (DR) are not supported.

Note

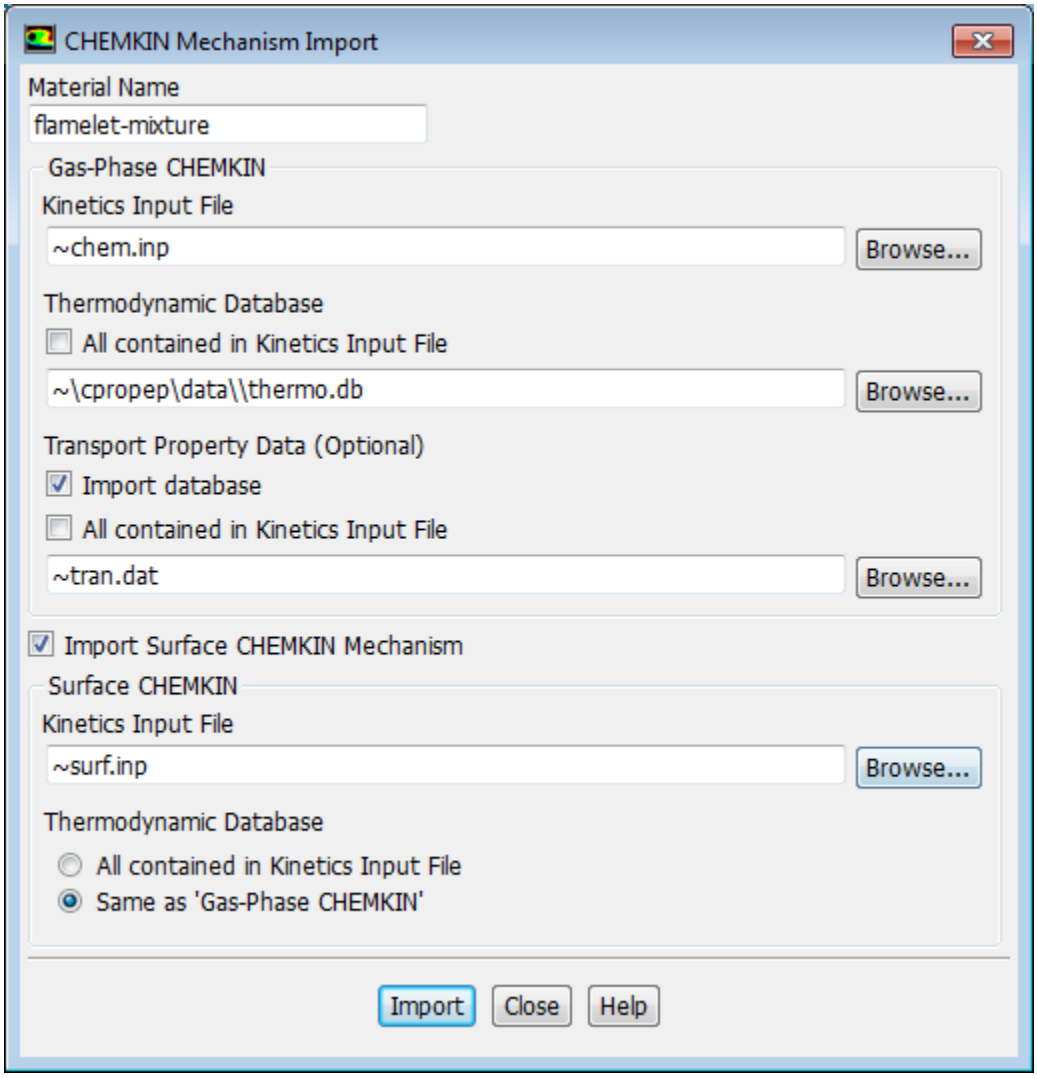
The proprietary reaction data will not be displayed in the Fluent interface.

14.1.3.2. Procedure for Importing Volumetric CHEMKIN Mechanisms

You can import the CHEMKIN mechanism file into ANSYS Fluent using the **CHEMKIN Mechanism Import** dialog box (Figure 14.5: The CHEMKIN Mechanism Import Dialog Box for Volumetric Kinetics (p. 969)) (opened by clicking **Import CHEMKIN Mechanism...** in the **Species Model** dialog box).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.5: The CHEMKIN Mechanism Import Dialog Box for Volumetric Kinetics



In the **CHEMKIN Mechanism Import** dialog box

1. Enter a name for the chemical mechanism under **Material Name**.

2. Enter the path to the CHEMKIN file (for example, *path/file*.inp) in the **Kinetics Input File** text field in the **Gas-Phase CHEMKIN** group box.

   Important

   Since ANSYS Fluent does not solve for the last species, you must ensure that the last species in the CHEMKIN mechanism species list is an abundant species, such as nitrogen. If not, edit the CHEMKIN mechanism file before importing it into ANSYS Fluent, and move the most abundant species (that is the species in your system with the largest total mass fraction) to the end of the species list.

3. Specify species thermodynamic property data. You can include the data in the kinetics input file and/or in a separate file as follows:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you want to specify thermodynamic data for all species in the kinetics input file, enable **All contained in Kinetics Input File** to avoid the need to specify a separate file.
- If you provide some or all species thermodynamic data in a separate file, disable **All contained in Kinetics Input File** and specify the thermodynamic data file location.

You can either use the default thermodynamic database file, `thermo.db`, provided in the `cpropep\data` directory in your ANSYS Fluent installation area or specify the path of your thermodynamic database file if `thermo.db` does not contain all the gas-phase species in the CHEMKIN mechanism. The format for `thermo.db` is detailed in the CHEMKIN manual [52] (p. 2889).

4. (Optional). To import transport property data, enable **Import database** under **Transport Property Data (Optional)** and specify the location of the species transport property data. You can include the data in the kinetics input file and/or in a separate file as follows:

   - If you want to specify transport property data for all species in the kinetics input file, enable **All contained in Kinetics Input File**, to avoid the need to specify a separate file.
   - If you want to provide some or all species transport data in a separate file, disable **All contained in Kinetics Input File** and specify the transport property data file location.

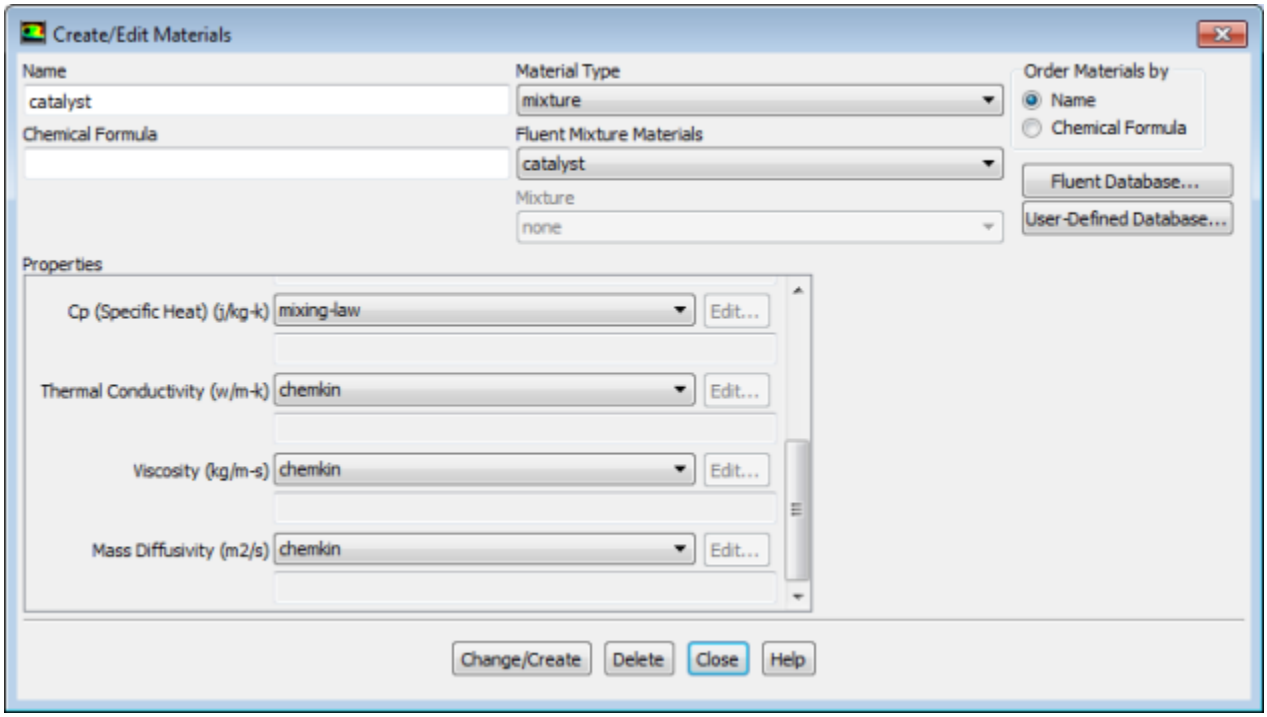
   ANSYS Fluent will enable mixture-averaged multicomponent diffusion and set the Lennard-Jones kinetic theory parameters for all the species in the imported CHEMKIN mechanism. If you would like to use Stefan-Maxwell diffusion, enable the **Full Multicomponent Diffusion** in the **Species** dialog box.

   If you want to use native CHEMKIN transport property calculations, you must first select **CHEMKIN-CFD Solver** from the **Chemistry Solver** drop-down list in the **Species** dialog box (Figure 14.1: The Species Model Dialog Box (p. 961)). Once a file with CHEMKIN transport properties is read, ANSYS Fluent will create a material with the specified name, which will contain the data for the species and reactions, and add it to the list of available **Mixture Materials** in the Create/Edit Materials Dialog Box (p. 2302). For material properties such as **Specific Heat**, **Viscosity**, **Thermal Conductivity**, **Mass Diffusivity**, and **Thermal Diffusion**, listed in the **Create/Edit Materials** dialog box, the option **chemkin** will appear in the drop-down list of each of the properties (Figure 14.6: The Material Dialog Box When Importing CHEMKIN Transport Properties (p. 971)), allowing you to use material property values computed by ANSYS Chemkin-Pro.

   Note that when **Full Multicomponent Diffusion** is enabled in the **Species** dialog box, ANSYS Chemkin-Pro returns full multicomponent diffusivities. If **Full Multicomponent Diffusion** is disabled, ANSYS Chemkin-Pro returns mixture averaged mass diffusivities for each species.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.6: The Material Dialog Box When Importing CHEMKIN Transport Properties



5. Click the **Import** button.

   Once you click **Import**, ANSYS Fluent automatically saves the reaction mechanism and thermal files in the data structure. The original input files are not required to be retained along with the Fluent case file. ANSYS Fluent will automatically regenerate these files as needed (for example, when using the CHEMKIN-CFD solver).

For information on how to import a surface kinetic mechanism in CHEMKIN format, see Importing a Surface Kinetic Mechanism in CHEMKIN Format (p. 995).

14.1.3.3. CHEMKIN Mechanisms Included with ANSYS Fluent

A CHEMKIN mechanism is a chemical reaction mechanism (also called a chemistry set) that has the following mechanism components:

- A file containing thermochemical data for the chemical species of interest
- A file containing a description of the reactions occurring in the gas phase
- (Optional) A file containing a description of the reactions occurring at the gas-surface interface or on the surface
- (Optional) A file containing gas-phase transport data

A set of gas-phase and/or surface reactions are generally developed using specific thermochemical data. Thus, the gas-phase kinetics, surface kinetics, and thermodynamic data are treated as a chemistry "set" in reacting-flow simulations. ANSYS Fluent provides the following fuel-combustion mechanisms (including reactions, thermodynamic data, and transport data) that are appropriate for use in common combustion simulations:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Methane / Ethane: 50-species version of GRI-Mech 3.0 (`grimech30_50spec`)

  This is the original `GRI-mech` 3.0 (53 species) [109] (p. 2892) with three species (argon AR, propane C3H8, and propyl radical C3H7) and their reactions removed.

  In fuel/air or fuel/oxygen combustion, the removal of these species is expected to have negligible effect on the mechanism predictive capabilities for methane and ethane as fuels. The applicability of the GRI mechanism and validation cases are provided on the GRI mechanism web site [109] (p. 2892).

- Propane: 37-species high-temperature mechanism (`Propane-NOx_highT`)

  The mechanism has 211 elementary reaction steps and was obtained by removing species and reactions that are unimportant for propane combustion from the comprehensive master mechanism [73] (p. 2891). ANSYS Chemkin-Pro Reaction Workbench software was used to reduce the master mechanism under the conditions of slightly lean and high pressure laminar flame of a propane/air mixture using the Directed Relation Graph (DRG) method.

- Two hydrogen mechanisms:

  – 9-species version for H2 combustion (`H2_mech`)

    The mechanism has 21 elementary reaction steps and has been extracted from the Li et al. [57] (p. 2890) mechanism for CO and other hydrocarbons.

  – 21-species version for H2 combustion with NOx (`H2_NOx_mech`)

    The mechanism has 105 elementary reaction steps. The hydrogen combustion sub-mechanism is the same as the `H2_mech` mechanism, above. The NO_x sub-mechanism is based on the study by the Glarborg group [91] (p. 2892).

14.1.4. Defining Properties for the Mixture and Its Constituent Species

As discussed in Overview of User Inputs for Modeling Species Transport and Reactions (p. 958), if you use a mixture material from the ANSYS Fluent database, most mixture and species properties will already be defined. You may follow the procedures in this section to check the current properties, modify some of the properties, or set all properties for a brand-new mixture material that you are defining from nothing.

Remember that you will need to define properties for the mixture material and also for its constituent species. It is important that you define the mixture properties before setting any properties for the constituent species, since the species property inputs may depend on the methods you use to define the properties of the mixture.

The recommended sequence for property inputs is as follows:

1. Define the mixture species, and reaction(s), and define physical properties for the mixture. Remember to click the **Change/Create** button when you are done setting properties for the mixture material.

2. Define physical properties for the species in the mixture. Remember to click the **Change/Create** button after defining the properties for each species.

These steps, all of which are performed in the Create/Edit Materials Dialog Box (p. 2302), are described in detail in this section.

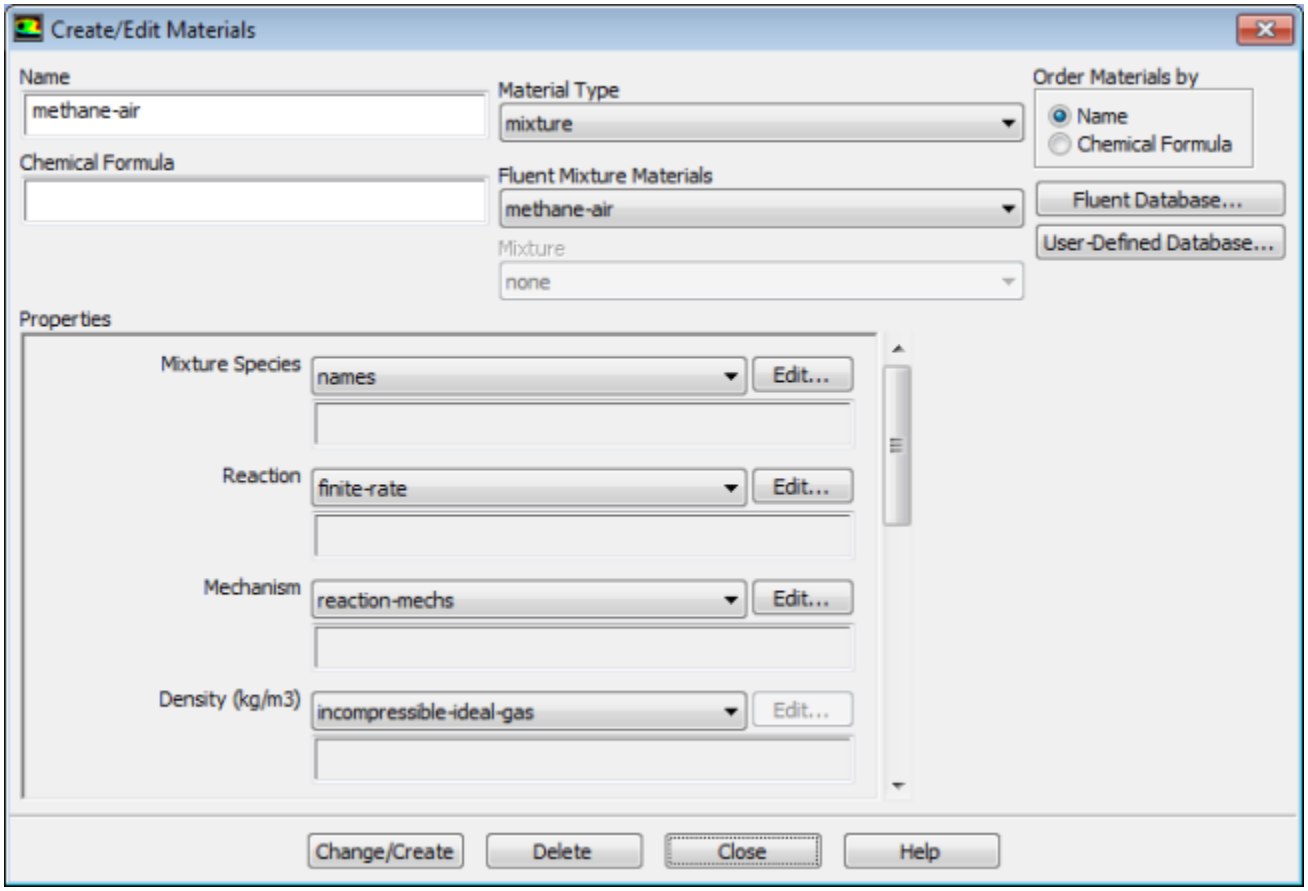
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → Materials

14.1.4.1. Defining the Species in the Mixture

In the **Create/Edit Materials** dialog box (Figure 14.7: The Create/Edit Materials Dialog Box (Showing a Mixture Material) (p. 973)), check that the **Material Type** is set to **mixture** and your mixture is selected in the **Fluent Mixture Materials** list.

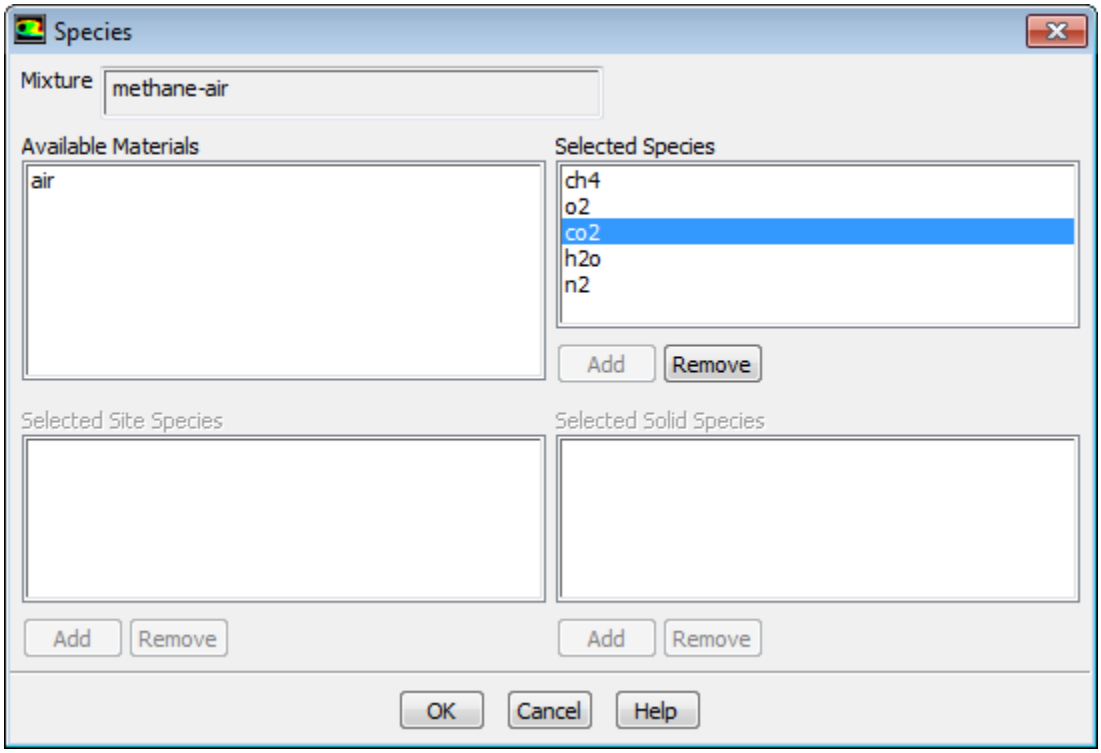
Figure 14.7: The Create/Edit Materials Dialog Box (Showing a Mixture Material)



Click the **Edit...** button to the right of **Mixture Species** under the **Properties** group box and in the Species Dialog Box (p. 2332) (Figure 14.8: The Species Dialog Box (p. 974)) that opens define the species in the mixture material.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.8: The Species Dialog Box



Important

For Chemkin mechanisms, the recommended approach for modifying mixture materials is to modify the Chemkin mechanism file and then re-import it into your Fluent case.

14.1.4.1.1. Overview of the Species Dialog Box

In the **Species** dialog box, the **Selected Species** list shows all of the fluid-phase species in the mixture. If you are modeling wall or particle surface reactions, the **Selected Solid Species** list will show all of the bulk solid species in the mixture. Solid species are species that are deposit to, or etch from, wall boundaries or discrete-phase particles (for example, Si(s)) and do not exist as fluid-phase species. If you are modeling wall surface reactions with site balancing, where species adsorb onto the wall surface, react, and then desorb off the surface, the **Selected Site Species** list will show all of the site species in the mixture.

The use of solid and site species with wall surface reactions is described in Wall Surface Reactions and Chemical Vapor Deposition (p. 995). See Particle Surface Reactions (p. 1001) for information about particle surface reactions.

Important

The order of the species in the **Selected Species** list is very important. ANSYS Fluent considers the last species in the list to be the bulk species. You should therefore be careful to retain the most abundant species (by mass) as the last species when you add species to or delete species from a mixture material.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Available Materials** list shows materials that are available but not in the mixture. Generally, you will see air in this list, since air is always available by default.

14.1.4.1.2. Adding Species to the Mixture

If you are creating a mixture from nothing or starting from an existing mixture and adding some missing species, you will first need to load the desired species from the database (or create them, if they are not present in the database) so that they will be available to the solver. You will need to close the **Species** dialog box before you begin, since it is a "modal" dialog box that will not allow you to do anything else when it is open. The procedure for adding species is as follows:

1. In the **Create/Edit Materials** dialog box, click the **Fluent Database...** button to open the **Fluent Database Materials** dialog box and copy the desired species, as described in Copying Materials from the ANSYS Fluent Database (p. 464). Remember that the constituent species of the mixture are fluid materials, so you should select **fluid** as the **Material Type** in the **Fluent Database Materials** dialog box to see the correct list of choices. Note that available solid and site species (for surface reactions) are also contained in the **fluid** list.

   Important

   If you do not see the species you are looking for in the database, you can create a new fluid material for that species, following the instructions in Creating a New Material (p. 466), and then continue with step 2, below.

2. Re-open the **Species** dialog box. You will see that the fluid materials you copied from the database (or created) are listed in the **Available Materials** list.

3. To add a species to the mixture, select it in the **Available Materials** list and click the **Add** button below the **Selected Species** list (or below the **Selected Site Species** or **Selected Solid Species** list, to define a site or solid species). The species will be added to the end of the relevant list and removed from the **Available Materials** list.

4. Repeat the previous step for all the desired species. When you are finished, click the **OK** button.

Important

Adding a species to the list will alter the order of the species. You should be sure that the last species in the list is the bulk species, and you should check all cell and boundary zone conditions, under-relaxation factors, and other solution parameters that you have set, as described in detail in the following sections.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.1.4.1.3. Removing Species from the Mixture

To remove a species from the mixture, simply select it in the **Selected Species** list (or the **Selected Site Species** or **Selected Solid Species** list) and click the **Remove** button below the list. The species will be removed from the list and added to the **Available Materials** list.

Important

- Removing a species from the list will alter the order of the species. You should be sure that the last species in the list is the bulk species, and you should check any cell zone or boundary conditions, under-relaxation factors, or other solution parameters that you have set, as described in detail in the following sections.
- Any species that is a reactant, product, or enhanced third-body in an active reaction cannot be deleted from the mixture material.

14.1.4.1.4. Reordering Species

If you find that the last species in the **Selected Species** list is not the most abundant species (as it should be), you will need to rearrange the species to obtain the proper order.

1. Remove the bulk species from the **Selected Species** list. It will now appear in the **Available Species** list.
2. Add the species back in again. It will automatically be placed at the end of the list.

14.1.4.1.5. The Naming and Ordering of Species

As discussed previously, you should retain the most abundant species as the last one in the **Selected Species** list when you add or remove species. Additional considerations you should be aware of when adding and deleting species are presented here.

There are three characteristics of a species that identify it to the solver: name, chemical formula, and position in the list of species in the **Species** dialog box. Changing these characteristics will have the following effects:

- You can change the **Name** of a species (using the Create/Edit Materials Dialog Box (p. 2302), as described in Renaming an Existing Material (p. 463)) without any consequences.
- You should *never* change the given **Chemical Formula** of a species.
- You will change the order of the species list if you add or remove any species. When this occurs, all cell zone or boundary conditions, solver parameters, and solution data for species will be reset to the default values. (Solution data, cell zone or boundary conditions, and solver parameters for other flow variables will not be affected.) Therefore, if you add or remove species you should take care to redefine species cell zone and boundary conditions and solution parameters for the newly defined problem. In addition, you should recognize that patched species concentrations or concentrations stored in any data file that was based on the original species ordering will be incompatible with the newly defined problem. You can use the data file as a starting guess, but you should be aware that the species concentrations in the data file may provide a poor initial guess for the newly defined model.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.1.4.2. Defining Reactions

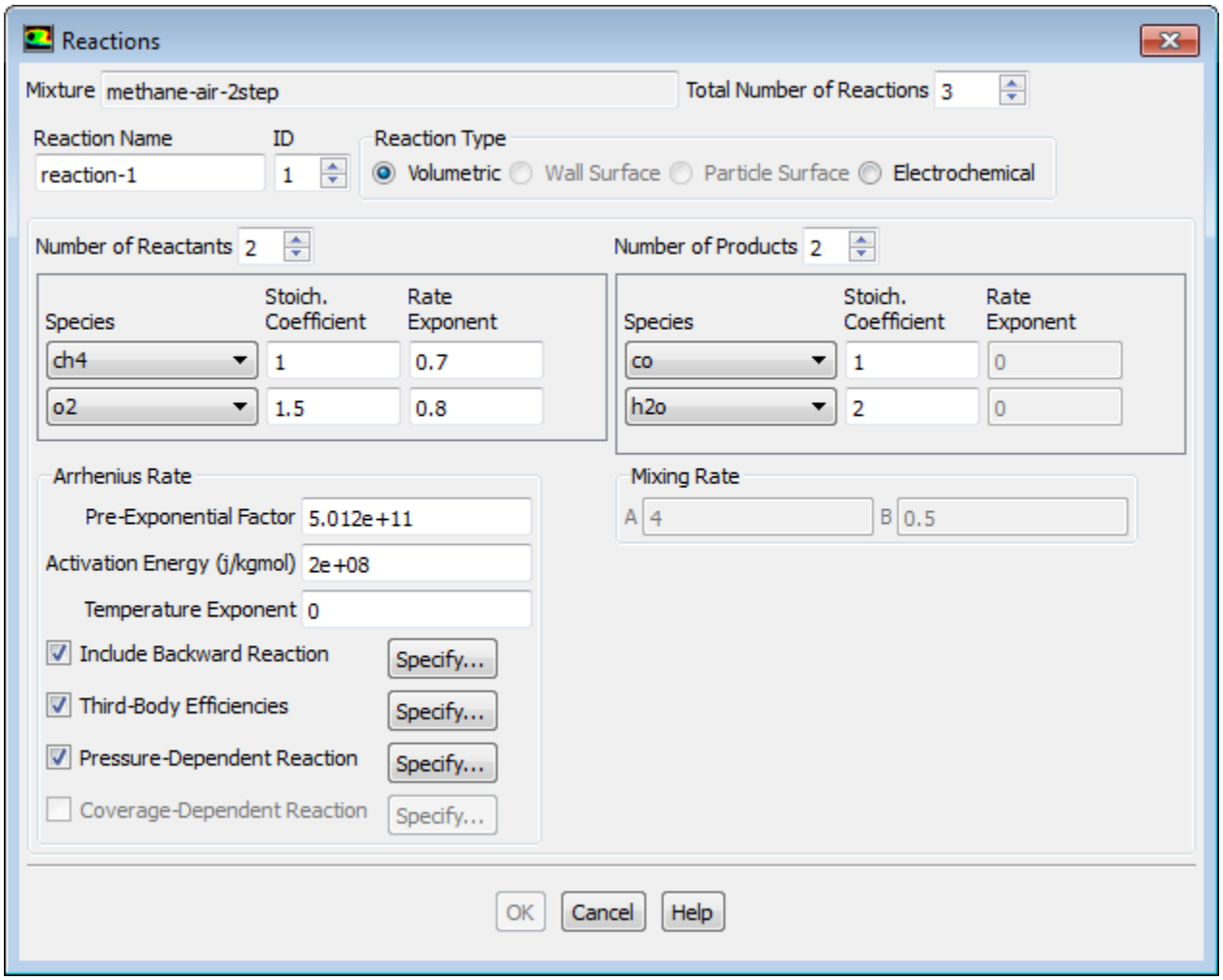
If your ANSYS Fluent model involves chemical reactions, you can define the reactions in which the defined species participate. This will be necessary only if you are creating a mixture material from nothing, you have modified the species, or you want to redefine the reactions for some other reason.

Depending on which turbulence-chemistry interaction model you selected in the **Species Model** dialog box (see Enabling Species Transport and Reactions and Choosing the Mixture Material (p. 960)), the appropriate reaction model will be displayed in the **Reaction** drop-down list in the **Edit Material** dialog box. If you are using the **Laminar Finite-Rate** or **Eddy-Dissipation Concept** model, the reaction model will be **finite-rate**; if you are using the **Eddy-Dissipation** model, the reaction model will be **eddy-dissipation**; if you are using the **Finite-Rate/Eddy-Dissipation** model, the reaction model will be **finite-rate/eddy-dissipation**.

14.1.4.2.1. Inputs for Reaction Definition

To define the reactions, click the **Edit...** button to the right of **Reaction**. The **Reactions** dialog box (Figure 14.9: The Reactions Dialog Box (p. 977)) will open.

Figure 14.9: The Reactions Dialog Box



The steps for defining reactions are as follows:

1. Set the total number of reactions (volumetric reactions, wall surface reactions, and particle surface reactions) in the **Total Number of Reactions** field. Use the arrows to change the value, or type in the value and press **Enter**.

   Note that if your model includes discrete-phase combusting particles, you should include the particulate surface reaction(s) (for example, char burnout, multiple char oxidation) in the number of reactions *only* if you plan to use the multiple surface reactions model for surface combustion.

2. Specify the **Reaction Name** of the reaction you want to define.

3. Set the **ID** of the reaction you want to define. Again, if you type in the value be sure to press **Enter**.

4. If this is a fluid-phase reaction, keep the default selection of **Volumetric** as the **Reaction Type**. If this is a wall surface reaction (described in Wall Surface Reactions and Chemical Vapor Deposition (p. 995)) or a particle surface reaction (described in Particle Surface Reactions (p. 1001)), select **Wall Surface** or **Particle Surface** as the **Reaction Type**. See User Inputs for Particle Surface Reactions (p. 1001) for further information about defining particle surface reactions.

5. Specify how many reactants and products are involved in the reaction by increasing the value of the **Number of Reactants** and the **Number of Products**. Select each reactant or product in the **Species** drop-down list and then set its stoichiometric coefficient and rate exponent in the appropriate **Stoich. Coefficient** and **Rate Exponent** fields. The stoichiometric coefficient is the constant $\nu'_{i,r}$ or $\nu''_{i,r}$ in Equation 7.6 in the Theory Guide and the rate exponent is the exponent on the reactant or product concentration, $\eta'_{j,r}$ or $\eta''_{j,r}$ in Equation 7.7 in the Theory Guide.

   There are two general classes of reactions that can be handled by the **Reactions** dialog box, so it is important that the parameters for each reaction are entered correctly. The classes of reactions are as follows:

   - Global forward reaction (no reverse reaction): Product species generally do not affect the forward rate, so the rate exponent for all products ($\eta''_{j,r}$) should be 0. For reactant species, set the rate exponent ($\eta'_{j,r}$) to the desired value. If such a reaction is not an elementary reaction, the rate exponent will generally not be equal to the stoichiometric coefficient ($\nu'_{i,r}$) for that species. An example of a global forward reaction is the combustion of methane:

     $$CH_4+2O_2\rightarrow CO_2+2H_2O \tag{14.1}$$

     where $\nu'_{CH_4}=1$, $\eta'_{CH_4}=0.2$, $\nu'_{O_2}=2$, $\eta'_{O_2}=1.3$, $\nu''_{CO_2}=1$, $\eta''_{CO_2}=0$, $\nu''_{H_2O}=2$, and $\eta''_{H_2O}=0$.

     Figure 14.9: The Reactions Dialog Box (p. 977) shows the coefficient inputs for the combustion of methane. See also the **methane-air** mixture material in the Fluent Database Materials Dialog Box (p. 2312).

     Note that, in certain cases, you may want to model a reaction where product species affect the forward rate. For such cases, set the product rate exponent ($\eta''_{j,r}$) to the desired value. An example of such a reaction is the gas-shift reaction (see the **carbon-monoxide-air** mixture material in the Fluent Database Materials Dialog Box (p. 2312)), in which the presence of water has an effect on the reaction rate:

     $$CO+\tfrac{1}{2}O_2+H_2O\rightarrow CO_2+H_2O$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the gas-shift reaction, the rate expression may be defined as:

$$k[CO][O_2]^{1/4}[H_2O]^{1/2} \tag{14.2}$$

where $\nu'_{CO}=1$, $\eta'_{CO}=1$, $\nu'_{O_2}=0.5$, $\eta'_{O_2}=0.25$, $\nu''_{CO_2}=1$, $\eta''_{CO_2}=0$, $\nu''_{H_2O}=0$, and $\eta''_{H_2O}=0.5$.

- Reversible reaction: An elementary chemical reaction that assumes the rate exponent for each species is equivalent to the stoichiometric coefficient for that species. An example of an elementary reaction is the oxidation of SO_2 to SO_3:

  $SO_2+\frac{1}{2}O_2 \rightleftharpoons SO_3$

  where $\nu'_{SO_2}=1$, $\eta'_{SO_2}=1$, $\nu'_{O_2}=0.5$, $\eta'_{O_2}=0.5$, $\nu''_{SO_3}=1$, and $\eta''_{SO_3}=1$.

  See step 7 below for information about how to enable reversible reactions.

6. If you are using the laminar finite-rate, finite-rate/eddy-dissipation, Eddy-Dissipation Concept or PDF Transport model for the turbulence-chemistry interaction, enter the following parameters for the Arrhenius rate in the **Arrhenius Rate** group box:

   Note

   All quantities for the Arrhenius rate parameters are in units of kgmol, m^3, and seconds.

   Pre-Exponential Factor

   (the constant A_r in Equation 7.9 in the Theory Guide). The units of A_r must be specified such that the units of the molar reaction rate, $\hat{R}_{i,r}$ in Equation 7.5 in the Theory Guide, are moles/volume-time (for example, kmol/m^3-s) and the units of the volumetric reaction rate, R_i in Equation 7.5 in the Theory Guide, are mass/volume-time (for example, kg/m^3-s).

   Important

   It is important to note that if you have selected the British units system, the Arrhenius factor should still be entered in SI units. This is because ANSYS Fluent applies no conversion factor to your input of A_r (the conversion factor is 1.0) when you work in British units, as the correct conversion factor depends on your inputs for $\nu'_{i,r}$, β_r, and so on.

   Activation Energy

   (the constant E_r in the forward rate constant expression, Equation 7.9 in the Theory Guide).

   Temperature Exponent

   (the value for the constant β_r in Equation 7.9 in the Theory Guide).

   Third-Body Efficiencies

   (the values for $\gamma_{j,r}$ in Equation 7.8 in the Theory Guide). If you have accurate data for the efficiencies and want to include this effect on the reaction rate (that is, include Γ in Equation 7.7 in the Theory Guide), select the **Third Body Efficiencies** option and click the **Specify...** button to open the Third-Body Efficiency Dialog Box (p. 2338) (Figure 14.10: The Third-Body Efficiency Dialog Box (p. 980)).

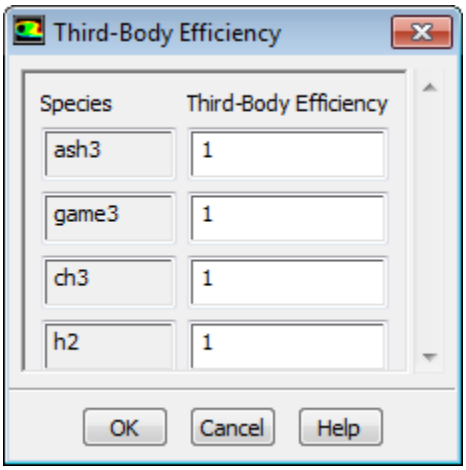
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For each **Species** in the dialog box, specify the **Third-Body Efficiency**.

Important

It is not necessary to include the third-body efficiencies. You should not enable the **Third-Body Efficiencies** option unless you have accurate data for these parameters.

Figure 14.10: The Third-Body Efficiency Dialog Box

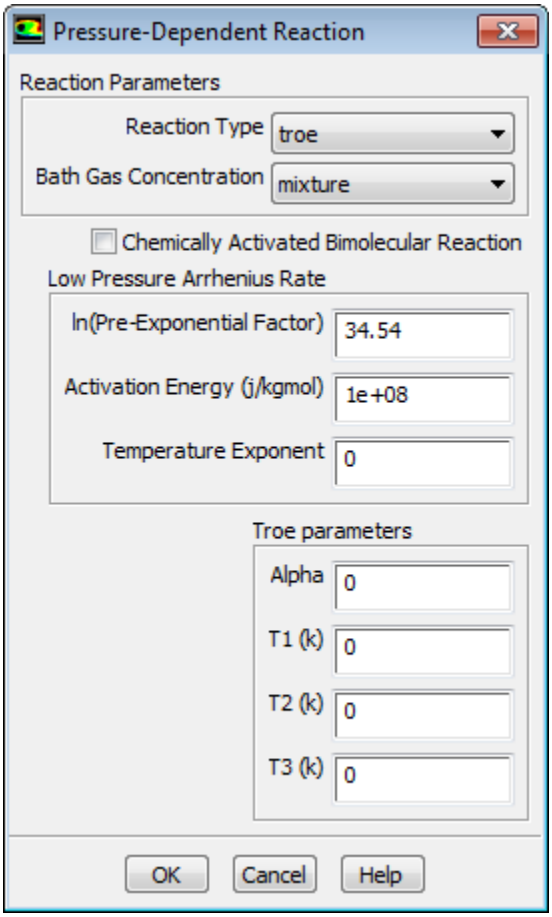


Pressure-Dependent Reaction

(if relevant) If you are using the laminar finite-rate or Eddy-Dissipation Concept model for turbulence-chemistry interaction, or have selected the composition PDF transport model (see Modeling a Composition PDF Transport Problem (p. 1113)), and the reaction is a pressure fall-off reaction (see Pressure-Dependent Reactions in the Theory Guide), enable the **Pressure-Dependent Reaction** option for the **Arrhenius Rate** and click the **Specify...** button to open the **Pressure-Dependent Reaction** dialog box (Figure 14.11: The Pressure-Dependent Reaction Dialog Box (p. 981)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.11: The Pressure-Dependent Reaction Dialog Box



Under **Reaction Parameters**, select the appropriate **Reaction Type** (**lindemann**, **troe**, or **sri**). See Pressure-Dependent Reactions in the Theory Guide for details about the three methods.

Next, specify if the **Bath Gas Concentration** ($[M]$ in Equation 7.18 in the Theory Guide) is to be defined as the concentration of the **mixture**, or as the concentration of one of the mixture's constituent species, by selecting the appropriate item in the drop-down list.

Enabling the **Chemically Activated Bimolecular Reaction** option results in a net rate constant at any pressure being defined as Equation 7.24 in the Theory Guide.

The parameters you specified under **Arrhenius Rate** in the **Reactions** dialog box represent the high-pressure Arrhenius parameters. You can, however, specify values for the following parameters under **Low Pressure Arrhenius Rate**:

ln(Pre-Exponential Factor)

(A_{low} in Equation 7.16 in the Theory Guide) The pre-exponential factor A_{low} is often an extremely large number, so you will input the natural logarithm of this term.

Activation Energy

(E_{low} in Equation 7.16 in the Theory Guide)

Temperature Exponent

(β_{low} in Equation 7.16 in the Theory Guide)

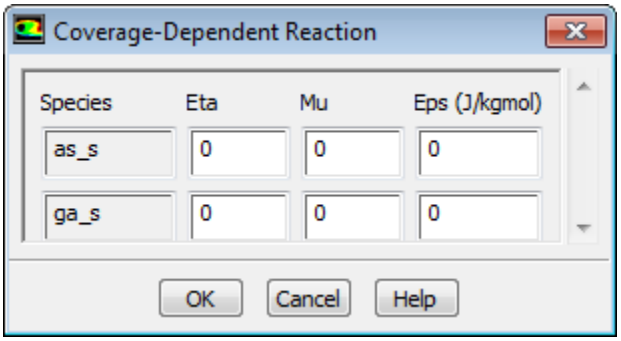
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you selected **troe** for the **Reaction Type**, you can specify values for **Alpha**, **T1**, **T2**, and **T3** (α, T_1, T_2, and T_3 in Equation 7.21 in the Theory Guide) under **Troe parameters**. If you selected **sri** for the **Reaction Type**, you can specify values for **a**, **b**, **c**, **d**, and **e** (a, b, c, d, and e in Equation 7.22 in the Theory Guide) under **SRI parameters**.

Coverage Dependent Reaction

If you are modeling **Wall Surface** reactions with site-balancing and you have reaction rates that depend on site coverages, you can enable the **Coverage Dependent Reaction** option. Click **Specify...** to open the **Coverage Dependent Reaction** dialog box (Figure 14.12: The Coverage Dependent Reaction Dialog Box (p. 982)) and input the coverage parameters.

Figure 14.12: The Coverage Dependent Reaction Dialog Box



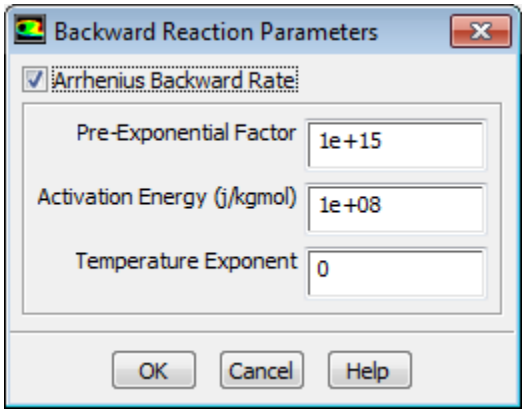
In the **Coverage Dependent Reaction** dialog box, all the site species of the reaction will be present with a default value of 0 for all the parameters, corresponding to no surface coverage modification. Enter the relevant values of the parameters μ, ε, and η (as defined in Equation 7.48 in the Theory Guide) for all the species for which the reaction has coverage dependence. μ and ε are dimensionless, and η is in units of J/kgmol.

7. If you are using the laminar finite-rate, Eddy-Dissipation Concept or PDF Transport model for turbulence-chemistry interaction, and the reaction is reversible, enable the **Include Backward Reaction** option for the **Arrhenius Rate**. When this option is enabled, you will not be able to edit the **Rate Exponent** for the product species, which instead will be set to be equivalent to the corresponding product **Stoich. Coefficient**. By default, ANSYS Fluent calculates the reverse reaction rate constants using Equation 7.10 in the *Fluent Theory Guide*. You can overwrite the ANSYS Fluent's default parameters for the reactions or define your own reactions. In both scenarios, you also need to specify the standard-state enthalpy and standard-state entropy for mixture material. ANSYS Fluent will use these values in the calculation of the backward reaction rate constant (Equation 7.10 in the *Fluent Theory Guide*).

   In addition, you have the option to provide your own backward reaction parameters. To do this, click **Specify...** next to **Include Backward Reaction** and enable the **Arrhenius Backward Rate** option in the **Backward Reaction Parameters** dialog box that opens (see Figure 14.13: Backward Reaction Parameters Dialog Box (p. 983)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.13: Backward Reaction Parameters Dialog Box



You can specify the following backward rate parameters:

Pre-Exponential Factor
(the constant $A_{b,r}$ in Equation 7.14 in the *Fluent Theory Guide*).

Activation Energy
(the constant $E_{b,r}$ in the backward rate constant expression in Equation 7.14 in the *Fluent Theory Guide*).

Temperature Exponent
(the constant $\beta_{b,r}$ in Equation 7.14 in the *Fluent Theory Guide*).

ANSYS Fluent will use your custom values to calculate the backward rate constant of the reversible reaction according to Equation 7.14 in the *Fluent Theory Guide*.

Note that the reversible reaction option is not available for either the eddy-dissipation or the finite-rate/eddy-dissipation turbulence-chemistry interaction model.

8. If you are using the eddy-dissipation or finite-rate/eddy-dissipation model for turbulence-chemistry interaction, you can enter values for **A** and **B** under the **Mixing Rate** heading. These values should not be changed unless you have reliable data. In most cases you will use the default values.

   A is the constant A in the turbulent mixing rate (Equation 7.25 and Equation 7.26 in the Theory Guide) when it is applied to a species that appears as a reactant in this reaction. The default setting of 4.0 is based on the empirically derived values given by Magnussen et al. [64] (p. 2890).

   B is the constant B in the turbulent mixing rate (Equation 7.26 in the Theory Guide) when it is applied to a species that appears as a product in this reaction. The default setting of 0.5 is based on the empirically derived values given by Magnussen et al. [64] (p. 2890).

9. Repeat steps 2–8 for each reaction you need to define. After defining all reactions, click **OK**.

14.1.4.2.2. Defining Species and Reactions for Fuel Mixtures

Quite often, combustion systems will include fuel that is not easily described as a pure species (such as CH_4 or C_2H_6). Complex hydrocarbons, including fuel oil or even wood chips, may be difficult to define in terms of such pure species. However, if you have available the heating value and the ultimate analysis (elemental composition) of the fuel, you can define an equivalent fuel species and an equivalent heat of formation for this fuel. Consider, for example, a fuel known to contain 50% C, 6% H, and 44% O by weight. Dividing by atomic weights, you can arrive at a "fuel" species with the molecular formula $C_{4.17}$

$H_6\ O_{2.75}$. You can start from a similar, existing species or create a species from nothing, and assign it a molecular weight of 100.04 kg/kmol (4.17 × 12 + 6 × 1 + 2.75 × 16). The chemical reaction would be considered to be

$$C_{4.17}H_6O_{2.75}+4.295O_2\rightarrow 4.17CO_2+3H_2O \tag{14.3}$$

You will need to set the appropriate stoichiometric coefficients for this reaction.

The heat of formation (or standard-state enthalpy) for the fuel species can be calculated from the known heating value ΔH since

$$\Delta H=\sum_{i=1}^{N}h_i^0\left(\nu_{i,r}''-\nu_{i,r}'\right) \tag{14.4}$$

where h_i^0 is the standard-state enthalpy on a molar basis. Note the sign convention in Equation 14.4 (p. 984) : ΔH is negative when the reaction is exothermic.

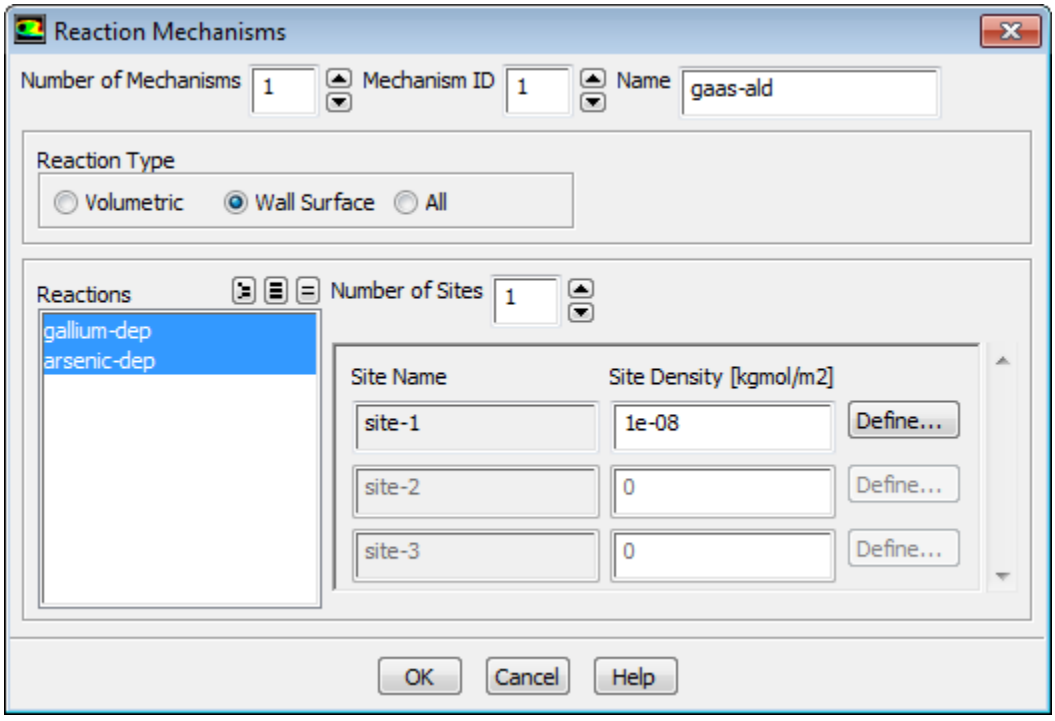
14.1.4.3. Defining Zone-Based Reaction Mechanisms

If your ANSYS Fluent model involves reactions that are confined to a specific area of the domain, you can define "reaction mechanisms" to enable different reactions selectively in different geometrical zones. You can create reaction mechanisms by selecting reactions from those defined in the **Reactions** dialog box and grouping them. You can then assign a particular mechanism to a particular zone.

14.1.4.3.1. Inputs for Reaction Mechanism Definition

To define a reaction mechanism, click the **Edit...** button to the right of **Mechanism**. The Reaction Mechanisms Dialog Box (p. 2342) (Figure 14.14: The Reaction Mechanisms Dialog Box (p. 984)) will open.

Figure 14.14: The Reaction Mechanisms Dialog Box



The steps for defining a reaction mechanism are as follows:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Set the total number of mechanisms in the **Number of Mechanisms** field. Use the arrows to change the value, or type the value and press **Enter**.

2. Set the **Mechanism ID** of the mechanism you want to define. Again, if you type in the value, be sure to press **Enter**.

3. Specify the **Name** of the mechanism.

4. Select the type of reaction to add to the mechanism under **Reaction Type**. If you select **Volumetric**, the **Reactions** list will display all available fluid-phase reactions. If you select **Wall Surface** or **Particle Surface**, the **Reactions** list will display all available wall surface reactions (described in Wall Surface Reactions and Chemical Vapor Deposition (p. 995)) or particle surface reactions (described in Particle Surface Reactions (p. 1001)). If you select **All**, the **Reactions** list will display all available reactions. This option is meant for backward compatibility with ANSYS Fluent 6.0 or earlier cases.

5. Select the reactions to be included in the mechanism.

   - For **Volumetric** or **Particle Surface** reactions, select available reactions for the mechanism in the **Reactions** list.

   - For **Wall Surface** reactions, use the following procedure:

     a. Select available wall surface reactions for the mechanism in the **Reactions** list.

     b. If any site species appear in the selected reaction(s), set the number of sites in the **Number of Sites** field. Use the arrows to change the value, or type the value and press **Enter**. See Reaction-Diffusion Balance for Surface Chemistry in the Theory Guide for details about site species in wall surface reactions.

     c. If you specify a **Number of Sites** that is greater than zero, specify the properties of the site.

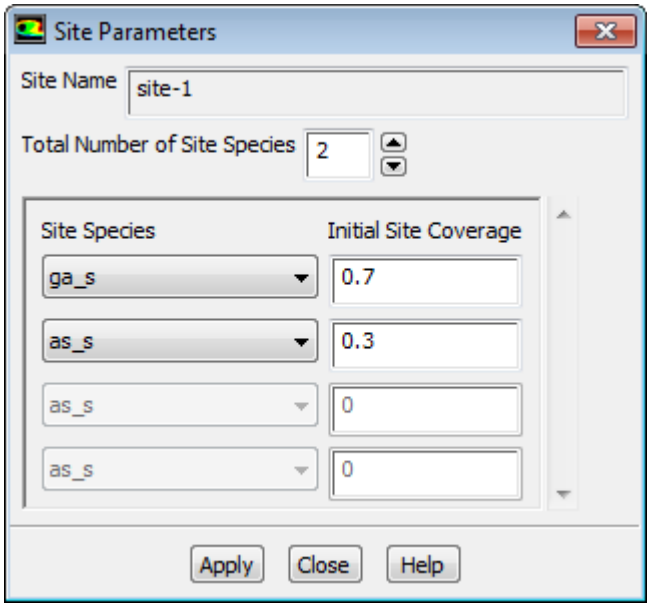
        Site Name
        (optional)

        Site Density
        (in kmol/m^2) This value is typically in the range of 10^{-8} to 10^{-6}.

        Click the **Define...** button. This will open the Site Parameters Dialog Box (p. 2344) (Figure 14.15: The Site Parameters Dialog Box (p. 986)), where you will define the parameters of the site species.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.15: The Site Parameters Dialog Box



Site Name

is the optional name of the site that was specified in the **Reaction Mechanisms** dialog box.

Total Number of Site Species

is the number of adsorbed species that are to be modeled at the site. (Use the arrows to change the value, or type the value and press **Enter**.)

Under **Site Species**, select the appropriate species from the drop-down list(s) and specify the fractional **Initial Site Coverage** for each species. For steady-state calculations, it is recommended (though not strictly required) that the initial values of **Initial Site Coverage** sum to unity. For transient calculations, it is required that these values sum to unity.

Click **Apply** in the **Site Parameters** dialog box to store the new values.

6. Repeat steps 2–5 for each reaction mechanism you need to define. When you are finished defining all reaction mechanisms, click **OK**.

14.1.4.4. Defining Physical Properties for the Mixture

When your ANSYS Fluent model includes chemical species, the following physical properties must be defined, either by you or by the database, for the mixture material:

- density, which you can define using the gas law or as a volume-weighted function of composition
- viscosity, which you can define as a function of composition
- thermal conductivity and specific heat (in problems involving solution of the energy equation), which you can define as functions of composition
- mass diffusion coefficients and Schmidt number, which govern the mass diffusion fluxes (Equation 7.2 and Equation 7.3 in the Theory Guide)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Detailed descriptions of these property inputs are provided in Physical Properties (p. 459).

Important

Remember to click the **Change/Create** button when you are done setting the properties of the mixture material. The properties that appear for each of the constituent species will depend on your settings for the properties of the mixture material. If, for example, you specify a composition-dependent viscosity for the mixture, you will need to define viscosity for each species.

14.1.4.5. Defining Physical Properties for the Species in the Mixture

For each of the fluid materials in the mixture, you (or the database) must define the following physical properties:

- molecular weight, which is used in the gas law and/or in the calculation of reaction rates and mole-fraction inputs or outputs
- standard-state (formation) enthalpy and reference temperature (in problems involving solution of the energy equation)
- viscosity, if you defined the viscosity of the mixture material as a function of composition
- thermal conductivity and specific heat (in problems involving solution of the energy equation), if you defined these properties of the mixture material as functions of composition
- standard-state entropy, if you are modeling reversible reactions
- thermal and momentum accommodation coefficients, if you have enabled the low-pressure boundary slip model.

Detailed descriptions of these property inputs are provided in Physical Properties (p. 459).

Important

Global reaction mechanisms with one or two steps inevitably neglect the intermediate species. In high-temperature flames, neglecting these dissociated species may cause the temperature to be overpredicted. A more realistic temperature field can be obtained by increasing the specific heat capacity for each species. Rose and Cooper [94] (p. 2892) have created a set of specific heat polynomials as a function of temperature.

The specific heat capacity for each species is calculated as

$$c_p(T) = \sum_{k=0}^{m} a_k T^k \tag{14.5}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The modified c_p polynomial coefficients (J/kg-K) are provided in Table 14.1: Modified Specific Heat Capacity (Cp) Polynomial Coefficients (p. 988).

Table 14.1: Modified Specific Heat Capacity (Cp) Polynomial Coefficients

	CO_2	H_2O	O_2
a_0	5.35446e+02	1.93780e+03	8.76317e+02
a_1	1.27867e+00	−1.18077e+00	1.22828e −01
a_2	−5.46776e −04	3.64357e −03	5.58304e −04
a_3	−2.38224e −07	−2.86327e −06	−1.20247e −06
a_4	1.89204e −10	7.59578e −10	1.14741e −09
a_5	—–	—–	−5.12377e −13
a_6	—–	—–	8.56597e −17

14.1.5. Setting up Coal Simulations with the Coal Calculator Dialog Box

The **Coal Calculator** dialog box automates calculation and setting of the relevant input parameters for the Species, Discrete-Phase (DPM) and Pollutant models associated with coal combustion. It is available in the **Species** dialog box for the **Species Transport** model when the **Eddy-Dissipation** or **Finite-Rate/Eddy-Dissipation** turbulence-chemistry option is selected. You can define up to three coal streams in your simulation using the **Number of Coal Streams** entry box.

For each coal stream selected in the **Coal Stream ID**, you need to define the following properties:

1. Coal **Proximate Analysis**, which is the mass fraction of **Volatile, Fixed Carbon, Ash** and **Moisture** in the coal. ANSYS Fluent will normalize the mass fractions so that they sum to unity.

2. Coal **Ultimate Analysis**, which is the mass fraction of atomic **C, H, O, N** and optionally **S**, in the Dry-Ash-Free (DAF) coal. ANSYS Fluent will normalize the mass fractions so that they sum to unity.

3. A choice of **One-step** or **Two-step** chemical mechanism. The one-step mechanism is,

$$vol + \nu_{O_2} O_2 \rightarrow \nu_{CO_2} CO_2 + \nu_{H_2O} H_2O + \nu_{SO_2} SO_2 + \nu_{N_2} N_2 \tag{14.6}$$

   The two-step mechanism involves oxidation of volatiles to CO in the first reaction and oxidation of CO to CO_2 in the second reaction:

$$\begin{aligned} vol + \nu_{O_2} O_2 &\rightarrow \nu_{CO} CO + \nu_{CO_2} CO_2 + \nu_{H_2O} H_2O + \nu_{SO_2} SO_2 + \nu_{N_2} N_2 \\ CO + 0.5 O_2 &\rightarrow CO_2 \end{aligned} \tag{14.7}$$

   The stoichiometric coefficients in Equation 14.6 (p. 988) and Equation 14.7 (p. 988) are calculated from the ultimate and proximate analyses.

4. An option to **Include SO2**. When this is enabled, an input for the atomic mass fraction of sulphur, **S**, appears in the ultimate analysis frame.

5. The **Coal Particle Material Name**. A DPM combusting-particle material will be created with this name. The default name is *coal-particle*.

6. The **Coal As-Received HCV**, where HCV denotes the Higher Calorific Value.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. **Volatile Molecular Weight** is the molecular weight of pure volatiles.

8. The **CO/CO2 Split in Reaction 1 Products** can be used to specify the molar fraction of CO to CO_2 in the first reaction of Equation 14.7 (p. 988). The default value of 1 implies that all carbon is reacted to CO, with no CO_2 produced.

9. The **High Temperature Volatile Yield**. Enhanced devolatization at higher temperatures can cause the volatile yield to exceed the proximate analysis fraction. To model this, the actual volatile fraction used is calculated as that specified in the **Proximate Analysis** input multiplied by the **High Temperature Volatile Yield**. The actual fixed carbon fraction is then calculated as one minus the sum of the actual volatile, ash and moisture fractions.

10. **Fraction of N in Char (DAF)**. This input is used in calculating the split of atomic nitrogen for the Fuel NOx model.

11. **Coal Dry Density** is used to calculate the volume fraction of liquid-water for the **Wet Combustion** option in the **Injections** dialog box.

Clicking **Apply** will save the **Coal Property** data for the selected **Coal Stream ID**. When you click **OK**, ANSYS Fluent makes the following changes:

1. A Mixture material is created, named *coal-volatiles-air*, with a one or two step reaction mechanism as specified in the **Mechanism** option. If the **Fluid** material species (O_2,CO,CO_2 and so on) do not exist, they are created. A **Fluid** material called *coal-volatiles*, is also created with a standard state enthalpy calculated from the ultimate and proximate analyses, as-received HCV and volatile molecular weight. For the second and third coal streams, additional **Fluid** materials and reactions with the properties corresponding to the second and third coal streams are also created in the mixture.

2. A combusting-particle material is created for each coal stream with **Volatile Component Fraction** and **Combustible Fraction** calculated from the ultimate and proximate analyses. The discrete phase model (DPM) is enabled.

3. For the fuel NOx model, the default fuel species is set to *vol*, the char N conversion is set to *NO*, and the fuel NOx **Volatile** and **Char** mass fractions are set according to the ultimate and proximate compositions of the first coal stream. Note that even though some of the default fuel NOx parameters are changed, the fuel NOx model itself is not enabled.

4. If the **Moisture** mass fraction in the coal proximate analysis is nonzero, then the **Wet Combustion** option is enabled in the **Coal Calculator** dialog box, and all subsequent injections that are created will have wet combustion enabled. The liquid material will be set to `water-liquid`, and the volume fraction of water will be calculated from the **Moisture** mass fraction specified in the proximate analysis and the **Coal Dry Density**. The wet combustion settings are not modified for existing injections. The **Density** for the combusting-particle in the **Create/Edit Materials** dialog box will also be set to **Coal Dry Density**.

Note

Irrespective of the **Wet Combustion** option in the **Coal Calculator** dialog box you can modify the **Wet Combustion** model setting for the injections in the **Set Injection Properties** dialog box. In addition, you will need to take care to enter the flow-rate of each injection on a wet or dry basis, consistent with the **Wet Combustion** model setting.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.1.6. Defining Cell Zone and Boundary Conditions for Species

You will need to specify the inlet mass fraction for all species in your simulation. In addition, for pressure outlets you will set species mass fractions to be used in case of backflow. At walls, ANSYS Fluent will apply a zero-gradient (zero-flux) boundary condition for all species by default, although you can change each species boundary condition to a specified value. If you have surface reactions defined (see Wall Surface Reactions and Chemical Vapor Deposition (p. 995)), you can choose to enable wall-surface reactions and select the chemical mechanism. For fluid zones, you also have the option of specifying a reaction mechanism. Input of cell zone and boundary conditions is described in Cell Zone and Boundary Conditions (p. 237).

Important

- Non-reflecting boundary conditions (NRBCs) are not compatible with species transport models. They are mainly used to solve ideal-gas single species flow. For information about NRBCs, see Boundary Acoustic Wave Models (p. 405).
- Note that you will explicitly set mass fractions only for the first $N-1$ species. The solver will compute the mass fraction of the last species by subtracting the total of the specified mass fractions from 1. If you want to explicitly specify the mass fraction of the last species, you must reorder the species in the list (in the Create/Edit Materials Dialog Box (p. 2302)), as described in Defining Properties for the Mixture and Its Constituent Species (p. 972).

14.1.6.1. Diffusion at Inlets with the Pressure-Based Solver

For the pressure-based solver in ANSYS Fluent, the net transport of species at inlets consists of both convection and diffusion components. The convection component is fixed by the specified inlet species mass or mole fraction, whereas the diffusion component depends on the gradient of the computed species concentration field (which is not known a priori). At very small convective inlet velocities, for example when modeling perforated combustion liners with an inlet, substantial mass can be gained or lost through the inlet due to diffusion. For this reason, inlet diffusion is disabled by default, but can be enabled with the **Inlet Diffusion** option in the **Species Model** dialog box.

Setup → Models → Species → Edit...

14.1.7. Defining Other Sources of Chemical Species

You can define a source or sink of a chemical species within the computational domain by defining a source term in the **Fluid** dialog box. You may choose this approach when species sources exist in your problem but you do not want to model them through the mechanism of chemical reactions. Defining Mass, Momentum, Energy, and Other Sources (p. 297) describes the procedures you would follow to define species sources in your ANSYS Fluent model. If the source is not a constant, you can use a user-defined function. See the Fluent Customization Manual for details about user-defined functions.

14.1.8. Solution Procedures for Chemical Mixing and Finite-Rate Chemistry

While many simulations involving chemical species may require no special procedures during the solution process, you may find that one or more of the solution techniques noted in this section helps to accelerate the convergence or improve the stability of more complex simulations. The techniques

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

outlined below may be of particular importance if your problem involves many species and/or chemical reactions, especially when modeling combusting flows.

14.1.8.1. Stability and Convergence in Reacting Flows

Obtaining a converged solution in a reacting flow can be difficult for a number of reasons. First, the impact of the chemical reaction on the basic flow pattern may be strong, leading to a model in which there is strong coupling between the mass/momentum balances and the species transport equations. This is especially true in combustion , where the reactions lead to a large heat release and subsequent density changes and large accelerations in the flow. All reacting systems have some degree of coupling, however, when the flow properties depend on the species concentrations. These coupling issues are best addressed by the use of a two-step solution process, as described below, and by the use of under-relaxation as described in Setting Under-Relaxation Factors (p. 1564).

A second convergence issue in reacting flows involves the magnitude of the reaction source term. When the ANSYS Fluent model involves very rapid reaction rates (reaction time scales are much faster than convection and diffusion time scales), the solution of the species transport equations becomes numerically difficult. Such systems are termed "stiff" systems. Stiff systems with finite-rate chemistry should be solved using either the pressure-based or density-based solver together with either **Stiff Chemistry Solver** or the **CHEMKIN-CFD Solver** (see Solution of Stiff Chemistry Systems (p. 992)). Finite-rate chemistry may also be used for turbulent combustion. For turbulent combustion, turbulence-chemistry interactions can either be neglected (**Finite-rate/No TCI** model) or accounted for using **Eddy-Dissipation Concept** or **PDF Transport** models.

14.1.8.2. Two-Step Solution Procedure (Steady-state Only)

Solving a reacting flow as a two-step process is a practical method for reaching a stable converged solution to your ANSYS Fluent problem. In this process, you begin by solving the flow, energy, and species equations with reactions disabled (the "cold-flow", or unreacting flow). When the basic flow pattern has thereby been established, you can re-enable the reactions and continue the calculation. The cold-flow solution provides a good starting solution for the calculation of the combusting system. This two-step approach to combustion modeling can be accomplished using the following procedure:

1. Set up the problem including all species and reactions of interest.

2. Temporarily disable reaction calculations by turning off **Volumetric** in the Species Model Dialog Box (p. 2216).

   Setup → **Models** → **Species** **Edit...**

3. Disable calculation of the product species in the Equations Dialog Box (p. 2511).

   Solution → **Controls** → **Equations...**

4. Calculate an initial (cold-flow) solution. (Note that it is generally not productive to obtain a fully converged cold-flow solution unless the non-reacting solution is also of interest to you.)

5. Enable the reaction calculations by turning on **Volumetric** again in the Species Model Dialog Box (p. 2216).

6. Enable all equations in the **Equations** dialog box. If you are using the **Finite-Rate/No TCI**, **Finite-Rate/Eddy-Dissipation**, **Eddy-Dissipation Concept** or **PDF Transport** model for the turbulence-chemistry interaction, you may need to patch an ignition source (as described in Ignition in Steady-State Combustion Simulations (p. 992)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.1.8.3. Density Under-Relaxation

One of the main reasons a combustion calculation can have difficulty converging is that large changes in temperature cause large changes in density, which can, in turn, cause instabilities in the flow solution. When you use the pressure-based solver, ANSYS Fluent allows you to under-relax the change in density to alleviate this difficulty. The default value for density under-relaxation is 1, but if you encounter convergence trouble you may want to reduce this to a value between 0.5 and 1 (in the **Solution Controls** task page).

Note that if you are using a real-gas model the solution might converge at a slower rate than when running ideal-gas flow. To improve the convergence performance of your analysis, you need to reduce the underrelaxation for density to values as low as 0.1.

14.1.8.4. Ignition in Steady-State Combustion Simulations

If you introduce fuel to an oxidant or have a premixed fuel/oxidant mixture, but you are not simulating the transient ignition process, then you must numerically force the solution to the burned state (rather than the unburned state). Generally, spontaneous ignition does not occur unless the temperature of the mixture exceeds the activation energy threshold required to maintain combustion. For steady-state ANSYS Fluent combustion simulations, whether you are using the species transport or PDF transport model and regardless of turbulence-chemistry interaction option, you have to supply an ignition source to initiate combustion. This ignition source may be a heated surface or inlet mass flow that heats the gas mixture above the required ignition temperature. Often, however, it is the equivalent of a spark: an initial solution state that causes combustion to proceed. You can supply this initial "spark" by patching a hot temperature into a region of the ANSYS Fluent model that contains an ignitable fuel/air mixture.

Solution → **Initialization** → **Patch...**

Depending on the model, you may need to patch both the temperature and the fuel/ oxidant/product concentrations to produce ignition in your model. The initial patch has no impact on the final steady-state solution—no more than the location of a match determines the final flow pattern of the torch that it lights. See Patching Values in Selected Cells (p. 1593) for details about patching initial values.

14.1.8.5. Solution of Stiff Chemistry Systems

When modeling stiff chemistry systems, such as flames with finite-rate kinetics, you can either use the pressure-based solver with the **Stiff Chemistry Solver** option enabled or **CHEMKIN-CFD Solver** selected as seen in the Species Model Dialog Box (p. 2216) (Figure 14.1: The Species Model Dialog Box (p. 961)), or the density-based solver.

For steady simulations using the pressure-based solver, the **Stiff Chemistry Solver** option approximates the reaction rate R_i in the species transport equation (see Equation 7.5 in the Theory Guide) as,

$$R_i^* = \frac{1}{\tau}\int_0^{\tau} R_i dt \tag{14.8}$$

where τ is an appropriate time-step. Note that as τ tends to zero the approximation becomes exact but the stiff numerics will cause the pressure-based solver to diverge. On the other hand, as τ tends to infinity, the approximated reaction rate R_i^* tends to zero and, while the numerical stiffness is alleviated, there is no reaction. In ANSYS Fluent, the default value for τ is set to one-tenth of the minimum convective or diffusive time-scale in the cell. This value was found to be sufficiently accurate and robust,

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

although it can be modified using the `solve/set/stiff-chemistry` text command. ISAT is employed to integrate the stiff chemistry in Equation 14.8 (p. 992). Equation 14.8 (p. 992) is also applied for unsteady simulations, with τ = flow time step.

Details about the ISAT algorithm may be found in Particle Reaction in the Theory Guide and Using ISAT Efficiently (p. 1130). For efficient and accurate use of ISAT, a review of this section is highly recommended.

The density-based implicit solver may also be used with either chemistry solver option. This option allows a larger stable Courant (CFL) number specification, although additional calculations are required to calculate the eigenvalues of the chemical system Jacobian [127] (p. 2893). When enabling the stiff-chemistry solver with the density-based solver, the following must be specified:

- **Temperature Positivity Rate Limit**: limits new temperature changes by this factor multiplied by the old temperature. Its default value is 0.2.
- **Temperature Time Step Reduction**: limits the local CFL number when the temperature is changing too rapidly. Its default value is 0.25.
- **Max. Chemical Time Step Ratio**: limits the local CFL number when the chemical time scales (eigenvalues of the chemical Jacobian) become too large to maintain a well-conditioned matrix. Its default value is 0.9.

14.1.8.6. Eddy-Dissipation Concept Model Solution Procedure

Due to the high computational expense of the Eddy-Dissipation Concept model, it is recommended that you use the following procedure to obtain a solution using the pressure-based solver:

1. Calculate an initial solution using the equilibrium Non-premixed or Partially-premixed model (see Modeling Non-Premixed Combustion (p. 1025) and Modeling Partially Premixed Combustion (p. 1099)).
2. Import a CHEMKIN format reaction mechanism (see Importing a Volumetric Kinetic Mechanism in CHEMKIN Format (p. 967)).
3. Enable the reaction calculations by turning on **Volumetric Reactions** in the **Species Model** dialog box and selecting **Eddy-Dissipation Concept** under **Turbulence-Chemistry Interaction**. Select the mechanism that you just imported as the **Mixture Material**.

   Setup → **Models** → **Species** **Edit...**
4. Set the species boundary conditions.

   Setup → **Boundary Conditions**
5. Disable the flow and turbulence and solve for the species and temperature only.
6. Enable all equations and iterate to convergence. Note that the default numerical parameters for the solution of the Eddy-Dissipation Concept equations are set to provide maximum robustness with slowest convergence. The convergence rate can be increased by setting the **Aggressiveness Factor** in the **Species** dialog box or with the text command:

   `define` → `models` → `species` → `set-turb-chem-interaction`

   The **Aggressiveness Factor** can be set from 0 (slow but stable) to 1 (fast but least stable). The default is 0.5

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.1.9. Postprocessing for Species Calculations

ANSYS Fluent can report chemical species as mass fractions, mole fractions, and molar concentrations. You can also display laminar and effective mass diffusion coefficients. The following variables are available for postprocessing of species transport and reaction simulations:

- **Mass fraction of species-n**
- **Mole fraction of species-n**
- **Molar Concentration of species-n**
- **Lam Diff Coef of species-n**
- **Eff Diff Coef of species-n**
- **Thermal Diff Coef of species-n**
- **Enthalpy of species-n** (pressure-based solver calculations only)
- **species-n Source Term** (density-based solver calculations only)
- **Relative Humidity**
- **TFM Thickening Factor** (Thickened Flame Model only)
- **TFM Omega** (Thickened Flame Model only)
- **TFM Thickening Factor** (turbulent cases (LES / DES / SAS / SBES / SDES) with Thickened Flame Model only)
- **Laminar Flame Speed** (Thickened Flame Model only)
- **Laminar Flame Thickness** (Thickened Flame Model only)
- **Cell Time Scale** (Eddy-Dissipation Concept and Laminar finite-rate stiff-chemistry only)
- **Fine Scale Mass fraction of species-n** (Eddy-Dissipation Concept model only)
- **EDC Cell Volume Fraction** (Eddy-Dissipation Concept model only)
- **Fine Scale Temperature** (Eddy-Dissipation Concept model only)
- **Net Rate of species-n** (Eddy-Dissipation Concept and Laminar finite-rate stiff-chemistry only)
- **Kinetic Rate of Reaction-n**
- **Turbulent Rate of Reaction-n**
- **Liquid species mass fraction of species-n** (solidification and melting model only)
- **Heat of Reaction**

These variables are contained in the **Species...**, **Temperature...**, and **Reactions...** categories of the variable selection drop-down list that appears in postprocessing dialog boxes. See Field Function Definitions (p. 1931) for a complete list of flow variables, field functions, and their definitions. Displaying

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Graphics (p. 1747) and Reporting Alphanumeric Data (p. 1885) explain how to generate graphics displays and reports of data.

14.1.9.1. Averaged Species Concentrations

Averaged species concentrations at inlets and exits, and across selected planes (that is, surfaces that you have created using **Create** in the **Setting Up Domain** ribbon tab (**Surface**group box)) within your model can be obtained using the Surface Integrals Dialog Box (p. 2616), as described in Surface Integration (p. 1920).

Results → **Reports** → **Surface Integrals** **Edit...**

Select the **Molar Concentration of species-n** for the appropriate species in the **Field Variable** drop-down list.

14.2. Wall Surface Reactions and Chemical Vapor Deposition

For gas-phase reactions, the reaction rate is defined on a volumetric basis and the rate of creation and destruction of chemical species becomes a source term in the species conservation equations. Heterogeneous surface reactions create sources (and sinks) of chemical species in the gas-phase as well, but also alter surface coverages for surface site reactions, and may deposit or etch species for bulk (solid) reactions.

For more information about the theoretical background of wall surface reactions and chemical vapor depositions, see Wall Surface Reactions and Chemical Vapor Deposition in the Theory Guide. Information about using wall surface reactions is presented in the following subsections:

14.2.1. Overview of Surface Species and Wall Surface Reactions
14.2.2. Importing a Surface Kinetic Mechanism in CHEMKIN Format
14.2.3. Manual Inputs for Wall Surface Reactions
14.2.4. Including Mass Transfer To Surfaces in Continuity
14.2.5. Wall Surface Mass Transfer Effects in the Energy Equation
14.2.6. Modeling the Heat Release Due to Wall Surface Reactions
14.2.7. Solution Procedures for Wall Surface Reactions
14.2.8. Postprocessing for Surface Reactions

14.2.1. Overview of Surface Species and Wall Surface Reactions

ANSYS Fluent treats chemical species adsorbed and desorbed, as well as those deposited into or etched from the bulk solid surfaces as distinct from the same chemical species in the gas. Similarly, surface reactions are defined distinctly and treated differently than gas-phase reactions involving the same chemical species.

Surface reactions can be limited so that they occur on only some of the wall boundaries (while the other wall boundaries remain free of surface reaction). The surface reaction rate is defined and computed per unit surface area, in contrast to the fluid-phase reactions, which are based on unit volume.

14.2.2. Importing a Surface Kinetic Mechanism in CHEMKIN Format

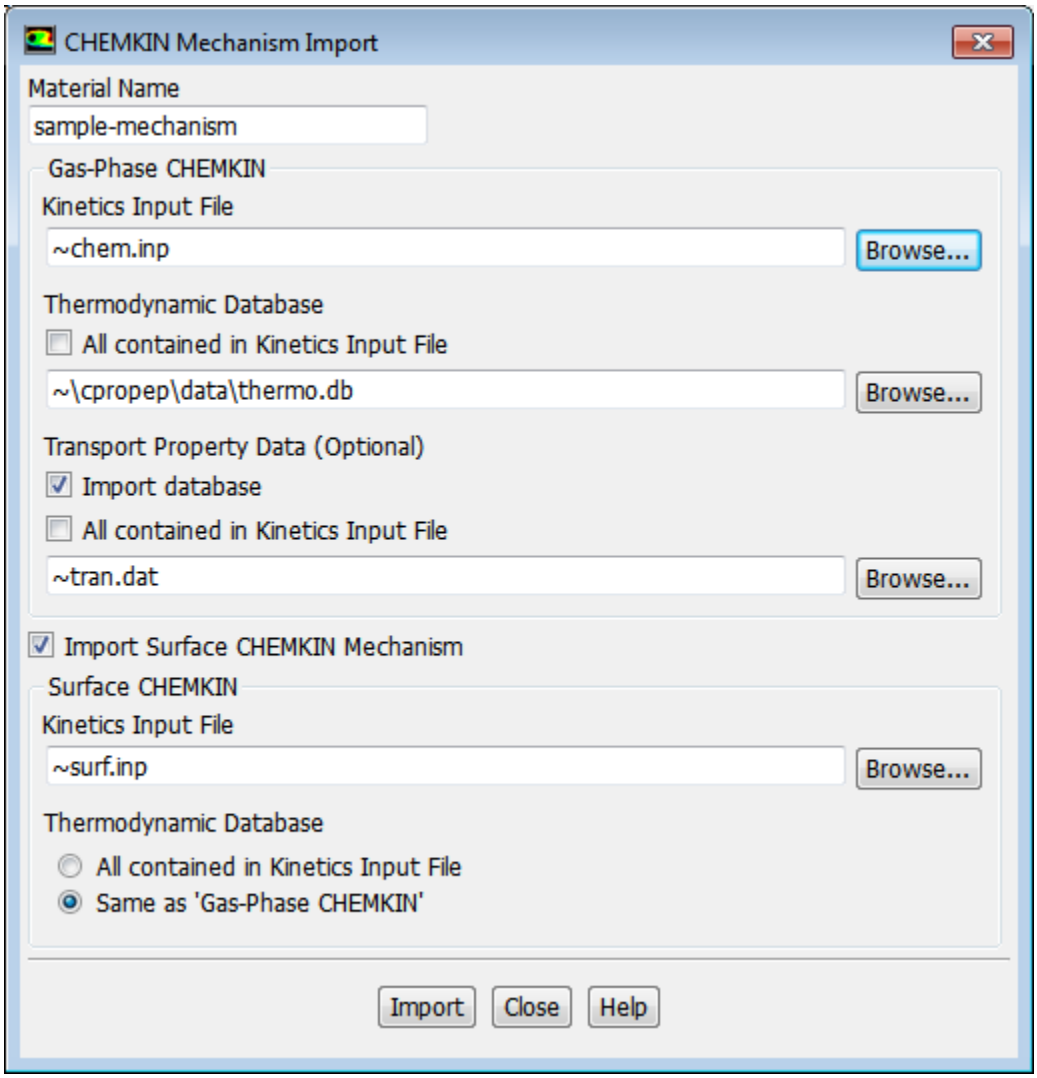
Importing surface kinetic mechanisms in CHEMKIN format requires that the gas-phase mechanism file accompanies the surface mechanism file for full compatibility with CHEMKIN (for more information on how to import the gas-phase mechanism, see Importing a Volumetric Kinetic Mechanism in CHEMKIN

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Format (p. 967)). If the gas-phase mechanism file is not available, then you will need to create one that you will import along with the surface mechanism file.

The mechanism files are imported into ANSYS Fluent using the **CHEMKIN Mechanism Import** dialog box (Figure 14.16: The CHEMKIN Mechanism Import Dialog Box for Surface Kinetics (p. 996)) which is opened by clicking **Import CHEMKIN Mechanism...** in the **Species Model** dialog box.

Figure 14.16: The CHEMKIN Mechanism Import Dialog Box for Surface Kinetics



In the **CHEMKIN Mechanism Import** dialog box:

1. Enter a name for the chemical mechanism under **Material Name**.

2. Enable **Import Surface CHEMKIN Mechanism**.

3. Enter the paths in the **Kinetic Input File** text fields in the **Gas-Phase CHEMKIN** and **Surface CHEMKIN** group boxes (for example, *path*`/chem.inp` and *path*/ *path*`/surf.inp`, respectively).

4. Specify species thermodynamic property data. You can include the data in the kinetics input file and/or in a separate file as follows:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you want to specify thermodynamic data for all species in the kinetics input file, enable **All contained in Kinetics Input File** to avoid the need to specify a separate file.
- If you provide some or all species thermodynamic data in a separate file, disable **All contained in Kinetics Input File** and specify the thermodynamic data file location.

You can either use the default thermodynamic database file, `thermo.db`, provided in the `cpropep\data` directory in your ANSYS Fluent installation area or specify the path of your thermodynamic database file if `thermo.db` does not contain all the gas-phase species in the CHEMKIN mechanism. The format for `thermo.db` is detailed in the CHEMKIN manual [52] (p. 2889).

5. To specify the surface species thermodynamic database, you can:
   - Select **All contained in Kinetics Input File** under the **Thermodynamic Database** and include the surface species thermodynamics data in the surface kinetics input file.
   - Select **Same as 'Gas-Phase CHEMKIN'** and include the surface species thermodynamics data in the thermodynamics data file used in the gas-phase mechanism. Note that the default `thermo.db` file contains only gas-phase species. If you want to use this file, you will need to supply a thermodynamic database that includes surface species defined in your simulation.

   Note

   The surface species thermodynamic data must be included either in the surface kinetics input file or in the gas-phase thermodynamics data file. If you do not import thermodynamics data as a separate file in the gas-phase CHEMKIN mechanism, then all surface thermodynamics data must be contained in the surface kinetics input file.

6. Click the **Import** button.

ANSYS Fluent will create a material with the specified name, which will contain the CHEMKIN data for the species and reactions, and add it to the list of **Fluent Mixture Materials**. You can view all of the reactions by clicking the **Edit...** button to the right of **Mechanism**, under **Properties** in the Create/Edit Materials Dialog Box (p. 2302).

ANSYS Fluent will also automatically save the imported data within the Fluent case file. Note that carrying the original input files along with the Fluent case file is not required. ANSYS Fluent will automatically retrieve these files when needed (for example, when using the CHEMKIN-CFD solver).

For details on importing gas species transport properties, see Procedure for Importing Volumetric CHEMKIN Mechanisms (p. 968).

Important

A surface reaction rate constant can be expressed in terms of a sticking coefficient in CHEMKIN mechanisms. When **Stiff Chemistry Solver** is selected from the **Chemistry Solver** drop-down list, ANSYS Fluent will convert this sticking coefficient form to an equivalent Arrhenius rate expression [51] (p. 2889). For full compatibility with ANSYS Chemkin-Pro, select **CHEMKIN-CFD Solver** instead.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.2.2.1. Compatibility and Limitations for Gas Phase Reactions

Generally, ANSYS Fluent handles all reaction types that are supported in ANSYS Chemkin-Pro. However, some limitations still exist for the following models and features:

- the density-based solver
- the **None – Explicit Source** chemistry solver (non-stiff chemistry)
- the **Finite-Rate/Eddy-Dissipation** turbulence-chemistry interaction

In these cases, only the following volume reaction types can be used:

- Arrhenius, including those reactions with arbitrary reaction order, third-body efficiencies and non-integer stoichiometric coefficients
- Pressure-dependent reactions using the Lindemann, Troe and SRI forms
- CHEMKIN mechanism files with CHEMKIN-default rate parameter units
- Duplicate reactions (keyword DUP)
- Arbitrary reverse reaction (keyword REV)

14.2.2.2. Compatibility and Limitations for Surface Reactions

As with volume reactions, ANSYS Fluent generally handles all reaction types that are supported in ANSYS Chemkin-Pro. However, limitations exist for the same models and features as described in Compatibility and Limitations for Gas Phase Reactions (p. 998). That is, with the density-based solver, **None – Explicit Source** chemistry solver, or **Finite-Rate/Eddy-Dissipation** turbulence-chemistry interaction, only the following surface reaction types can be used:

- Arrhenius reactions, including those with arbitrary reaction order, third-body efficiencies and non-integer stoichiometric coefficients
- Sticking coefficients (keyword STICK). ANSYS Fluent converts these to an equivalent Arrhenius expression.
- Arbitrary reaction units
- Duplicate reactions (keyword DUP)
- Surface coverage modification (keyword COV)

For a detailed description of the keywords, see ANSYS Chemkin-Pro Input Manual on the ANSYS Customer Portal (support.ansys.com/docdownloads).

14.2.3. Manual Inputs for Wall Surface Reactions

The basic steps for setting up a problem involving wall surface reactions are the same as those presented in Overview of User Inputs for Modeling Species Transport and Reactions (p. 958) for setting up a problem with only fluid-phase reactions, with a few additions:

1. In the Species Model Dialog Box (p. 2216):

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → Models → Species Edit...

a. Enable **Species Transport**, select **Volumetric** and **Wall Surface** under **Reactions**, and specify the **Mixture Material**. See Enabling Species Transport and Reactions and Choosing the Mixture Material (p. 960) for details about this procedure, and Mixture Materials (p. 959) for an explanation of the mixture material concept.

b. (optional) If you want to model the heat release due to wall surface reactions, enable the **Heat of Surface Reactions** option.

c. (optional) If you want to include the effect of surface mass transfer in the continuity equation, enable the **Mass Deposition Source** option.

d. To control the robustness and the convergence speed, enter a value between 0 and 1 for the **Aggressiveness Factor**. A value of 0 is the most robust, but results in the slowest convergence. The default value for the **Aggressiveness Factor** is 0.5.

e. (optional) If you are using the pressure-based solver and you want to include species diffusion effects in the energy equation, enable the **Diffusion Energy Source** option. See Wall Surface Mass Transfer Effects in the Energy Equation (p. 1000) for details.

f. (optional, but recommended for CVD) If you want to model full multicomponent (Stefan-Maxwell) diffusion or thermal (Soret) diffusion, enable the **Full Multicomponent Diffusion** or **Thermal Diffusion** option. See Full Multicomponent Diffusion (p. 520) for details.

2. Check and/or define the properties of the mixture. See Defining Properties for the Mixture and Its Constituent Species (p. 972).

   Setup → Materials

   Mixture properties include the following:

   - species in the mixture
   - reactions
   - other physical properties (for example, viscosity, specific heat)

   Important

   - You will find all species (including the solid/bulk and site species) in the list of **Fluent Fluid Materials**. For a deposited species such as Si, you will need both Si(g) and Si(s) in the materials list for the **fluid** material type.
   - Note that the *final* gas-phase species named in the **Selected Species** list should be the carrier gas. This is because ANSYS Fluent will not solve the transport equation for the final species. Note also that any reordering, adding or deleting of species should be handled with caution, as described in Reordering Species (p. 976).

3. Check and/or set the properties of the individual species in the mixture. (See Defining Properties for the Mixture and Its Constituent Species (p. 972).) Note that if you are modeling the heat of surface reactions, you should be sure to check (or define) the formation enthalpy for each species.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Set species boundary conditions.

   Setup → Boundary Conditions

   In addition to the boundary conditions described in Defining Cell Zone and Boundary Conditions for Species (p. 990), you will first need to indicate whether or not surface reactions are in effect on each wall. If so, you will then need to assign a reaction mechanism to the wall. To enable the effect of surface reaction on a wall, enable the **Reaction** option in the **Species** section of the **Wall** dialog box.

   Important

   If you have enabled the global **Low-Pressure Boundary Slip** option in the Viscous Model Dialog Box (p. 2176), the **Shear Condition** for each wall will be reset to **No Slip** even though the slip model will be in effect. Note that the **Low-Pressure Boundary Slip** option is available only when the **Laminar** model is selected in the **Viscous Model** dialog box.

   See Inputs at Wall Boundaries (p. 358) for details about boundary condition inputs for walls. See User Inputs for Porous Media (p. 267) for details about boundary condition inputs for porous media.

14.2.4. Including Mass Transfer To Surfaces in Continuity

In the surface reaction boundary condition described above, the effects of the wall normal velocity or bulk mass transfer to the wall are not included in the computation of species transport. The momentum of the net surface mass flux from the surface is also ignored because the momentum flux through the surface is usually small in comparison with the momentum of the flow in the cells adjacent to the surface. However, you can include the effect of surface mass transfer in the continuity equation by activating the **Mass Deposition Source** option in the Species Model Dialog Box (p. 2216).

14.2.5. Wall Surface Mass Transfer Effects in the Energy Equation

Species diffusion effects in the energy equation due to wall surface reactions are included in the normal species diffusion term described in Treatment of Species Transport in the Energy Equation in the Theory Guide.

If you are using the pressure-based solver, you can neglect this term by disabling the **Diffusion Energy Source** option in the Species Model Dialog Box (p. 2216). For the density-based solvers, this term is always included; you cannot disable it. Neglecting the species diffusion term implies that errors may be introduced to the prediction of temperature in problems involving mixing of species with significantly different heat capacities, especially for components with a Lewis number far from unity. While the effect of species diffusion should go to zero at Le = 1, you may see subtle effects due to differences in the numerical integration in the species and energy equations.

14.2.6. Modeling the Heat Release Due to Wall Surface Reactions

The heat release due to a wall surface reaction is, by default, ignored by ANSYS Fluent. You can, however, choose to include the heat of surface reaction by activating the **Heat of Surface Reactions** option in the Species Model Dialog Box (p. 2216) and setting appropriate formation enthalpies in the Edit Material Dialog Box (p. 2355).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.2.7. Solution Procedures for Wall Surface Reactions

As in all CFD simulations, your surface reaction modeling effort may be more successful if you start with a simple problem description, adding complexity as the solution proceeds. For wall surface reactions, you can follow the same guidelines presented for fluid-phase reactions in Solution Procedures for Chemical Mixing and Finite-Rate Chemistry (p. 990).

In addition, if you are modeling the heat release due to surface reactions and you are having convergence trouble, you should try temporarily turning off the **Heat of Surface Reactions** and **Mass Deposition Source** options in the Species Model Dialog Box (p. 2216).

If you are modeling surface site species, good estimates of the **Initial Site Coverage** will aid convergence.

14.2.8. Postprocessing for Surface Reactions

In addition to the gas-phase variables listed in Postprocessing for Species Calculations (p. 994), for surface reactions you can display/report the surface coverage as well as the deposition rate of the solid species deposited on a surface. Select **Surface Coverage of species-n**, **Surface Deposition Rate of species-n**, or **Porous Deposition Rate of species-n** in the **Species...** category of the variable selection drop-down list.

Important

For surface reactions involving porous media, you can display/report the surface reaction rates using the **Kinetic Rate of Reaction-n(Porous)** in the **Reactions...** category of the variable selection drop-down list.

14.3. Particle Surface Reactions

As described in The Multiple Surface Reactions Model in the Theory Guide, it is possible to define multiple particle surface reactions to model the surface combustion of a combusting discrete-phase particle. For more information about the theoretical background of particle surface reactions, see Particle Surface Reactions in the Theory Guide. Information about using particle surface reactions is provided in the following subsections:

14.3.1. User Inputs for Particle Surface Reactions
14.3.2. Modeling Gaseous Solid Catalyzed Reactions
14.3.3. Using the Multiple Surface Reactions Model for Discrete-Phase Particle Combustion

14.3.1. User Inputs for Particle Surface Reactions

The setup procedure for particle surface reactions requires only a few inputs in addition to the procedure for volumetric reactions described in Overview of User Inputs for Modeling Species Transport and Reactions (p. 958) – Defining Other Sources of Chemical Species (p. 990). These additional inputs are as follows:

- In the **Species Model** dialog box, enable the **Particle Surface** option under **Reactions**.

Setup → **Models** → **Species** **Edit...**

- When you specify the species involved in the particle surface reaction, be sure to identify the surface species, as described in Defining Properties for the Mixture and Its Constituent Species (p. 972).

  Important

  You will find all species (including the surface species) in the list of **Fluent Fluid Materials**. If, for example, you are modeling coal gasification, you will find solid carbon, C(s), in the materials list for the **fluid** material type.

- For each particle surface reaction, select **Particle Surface** as the **Reaction Type** in the **Reactions** dialog box, and specify the following parameters (in addition to those described in Defining Reactions (p. 977)):

  Diffusion-Limited Species

  When there is more than one gaseous reactant taking part in the particle surface reaction, the diffusion-limited species is the species for which the concentration gradient between the bulk and the particle surface is the largest. See Figure 7.1: A Reacting Particle in the Multiple Surface Reactions Model in the Theory Guide for an illustration of this concept. In most cases, there is a single gas-phase reactant and the diffusion-limited species does not need to be defined.

  Catalyst Species

  This option is available only when there are no solid species defined in the stoichiometry of the particle surface reaction. In such a case, you will need to specify the solid species that acts as a catalyst for the reaction. The reaction will proceed only on the particles that contain this solid species. See Using the Multiple Surface Reactions Model for Discrete-Phase Particle Combustion (p. 1002) for details on defining the particle surface species mass fractions.

  Diffusion Rate Constant

  ($C_{1,r}$ in Equation 7.71 in the Theory Guide)

  Effectiveness Factor

  (η_r in Equation 7.69 in the Theory Guide)

14.3.2. Modeling Gaseous Solid Catalyzed Reactions

The catalytic particle surface reaction option is enabled in ANSYS Fluent when **Particle Surface** is selected as the **Reaction Type** in the Reactions Dialog Box (p. 2334) and there are no solid species in the reaction stoichiometry. The solid species acting as a catalyst for the reaction is defined in the **Reactions** dialog box. The catalytic particle surface reaction will proceed only on those particles containing the catalyst species.

14.3.3. Using the Multiple Surface Reactions Model for Discrete-Phase Particle Combustion

When you use the multiple surface reactions model, the procedure for setting up a problem involving a discrete phase is slightly different from that outlined in Steps for Using the Discrete Phase Models (p. 1240). The revised procedure is as follows:

1. Enable any of the discrete phase modeling options, if relevant, as described in Physical Models for the Discrete Phase Model (p. 1249).

2. Specify the initial conditions, as described in Setting Initial Conditions for the Discrete Phase (p. 1265).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Define the boundary conditions, as described in Setting Boundary Conditions for the Discrete Phase (p. 1301).

4. Define the material properties, as described in Setting Material Properties for the Discrete Phase (p. 1313).

   Important

   You must select **multiple-surface-reactions** in the **Combustion Model** drop-down list in the **Create/Edit Materials** dialog box before you can proceed to the next step.

5. If you have defined more than one particle surface species, for example, carbon (**C<s>**) and sulfur (**S<s>**), you will need to return to the **Set Injection Properties** dialog box (or **Set Multiple Injection Properties** dialog box) to specify the mass fraction of each particle surface species in the combusting particle. Click the **Multiple Reactions** tab, and enter the **Species Mass Fractions**. These mass fractions refer to the combustible fraction of the combusting particle, and should sum to 1. If there is only one surface species in the mixture material, the mass fraction of that species will be set to 1, and you will not specify anything under **Multiple Surface Reactions**.

6. Set the solution parameters and solve the problem, as described in Solution Strategies for the Discrete Phase (p. 1325).

7. Examine the results, as described in Postprocessing for the Discrete Phase (p. 1329).

   Important

   Solid deposition reactions on the particle are not allowed together with custom laws.

14.4. Electrochemical Reactions

Similar to wall surface reactions, electrochemical reactions generate sources (and sinks) in chemical species transport equations.

For more information about the theoretical background of electrochemical reactions, see Electrochemical Reactions in the *Fluent Theory Guide*. Information about using electrochemical reactions is presented in the following sections:

14.4.1. Overview of Electrochemical Reactions
14.4.2. User Inputs for Electrochemical Reactions
14.4.3. Electrochemical Reaction Effects in the Energy Equation
14.4.4. Electrochemical Reaction Effects in the Species Transport Equation
14.4.5. Including Mass Transfer in Continuity
14.4.6. Solution Procedures for Electrochemical Reactions

14.4.1. Overview of Electrochemical Reactions

An electrochemical reaction occurs only on wall surfaces where the electrolyte and the electrode meet. Both aqueous and solid species can participate in electrochemical reactions. Solid species that can be involved in electrochemical reactions are corroded or deposited species.

Electrochemical reactions can be limited so that they can only occur on some of the wall boundaries (while the other wall boundaries remain free of reaction). The reaction rate is defined and computed per unit surface area, in contrast to the fluid-phase reactions, which are based on unit volume.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can specify different reaction mechanisms for different walls. Each reaction mechanism can have multiple electrochemical reactions.

14.4.2. User Inputs for Electrochemical Reactions

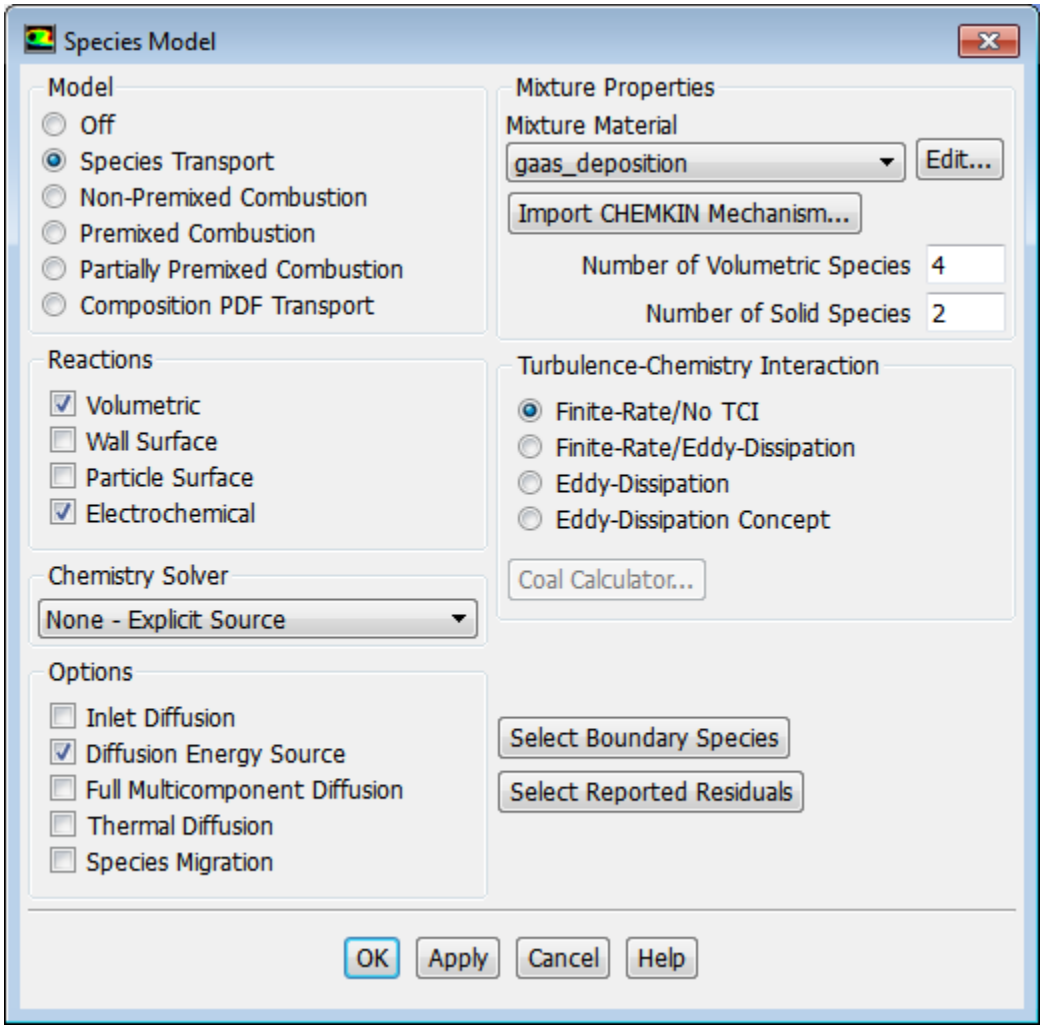
The steps needed to incorporate electrochemical reactions in your problem are as follows:

1. For unsteady simulations, select **Transient** from the **Setting Up Physics** ribbon tab (**Solver** group).

2. Enable chemical species transport model.

   Setup → Models → Species **Edit...**

   Figure 14.17: The Species Model Dialog Box with Electrochemical Reactions Enabled



   a. In the **Species Model** dialog box, under **Model**, enable **Species Transport**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Under **Reactions**, select **Volumetric** and **Electrochemical**.

> **Note**
>
> Once you enable the **Electrochemical** reactions, the electric potential solver will be automatically enabled (under the **Models** tree branch).

3. In the **Create/Edit Materials** dialog box, define all the species materials that are present in the electrolyte and electrodes.

Setup → **Materials**

a. Copy the new species from the Fluent database to your case as described in Copying Materials from the ANSYS Fluent Database (p. 464). If your species material is not present in the database, copy a similar species material and edit its properties, including **Name** and **Chemical Formula**.

> **Note**
>
> - When copying species (including corroded and deposited solid species), make sure that **fluid** is selected as **Material Type** in the **Fluent Database Materials** dialog box.
> - It is not necessary to represent the electron in the reaction, and e^- can be safely ignored.

b. Under the **Properties** group box, set **Charge Number** for the new species.

> **Important**
>
> By default, **Charge Number** is set to 0, which means that the species is neutral. Make sure to specify the **Charge Number** for all relevant species.

> **Note**
>
> Do not specify **Electrical Conductivity** for individual species. You will specify **Electrical Conductivity** for the mixture.

4. Edit the mixture composition using the **Species** dialog box (opened by clicking **Edit** next to **Mixture Species** under **Properties** of the **Create/Edit Materials** dialog box).

a. Remove unused species and add the electrochemical species to the **Selected Species** list as described in Adding Species to the Mixture (p. 975).

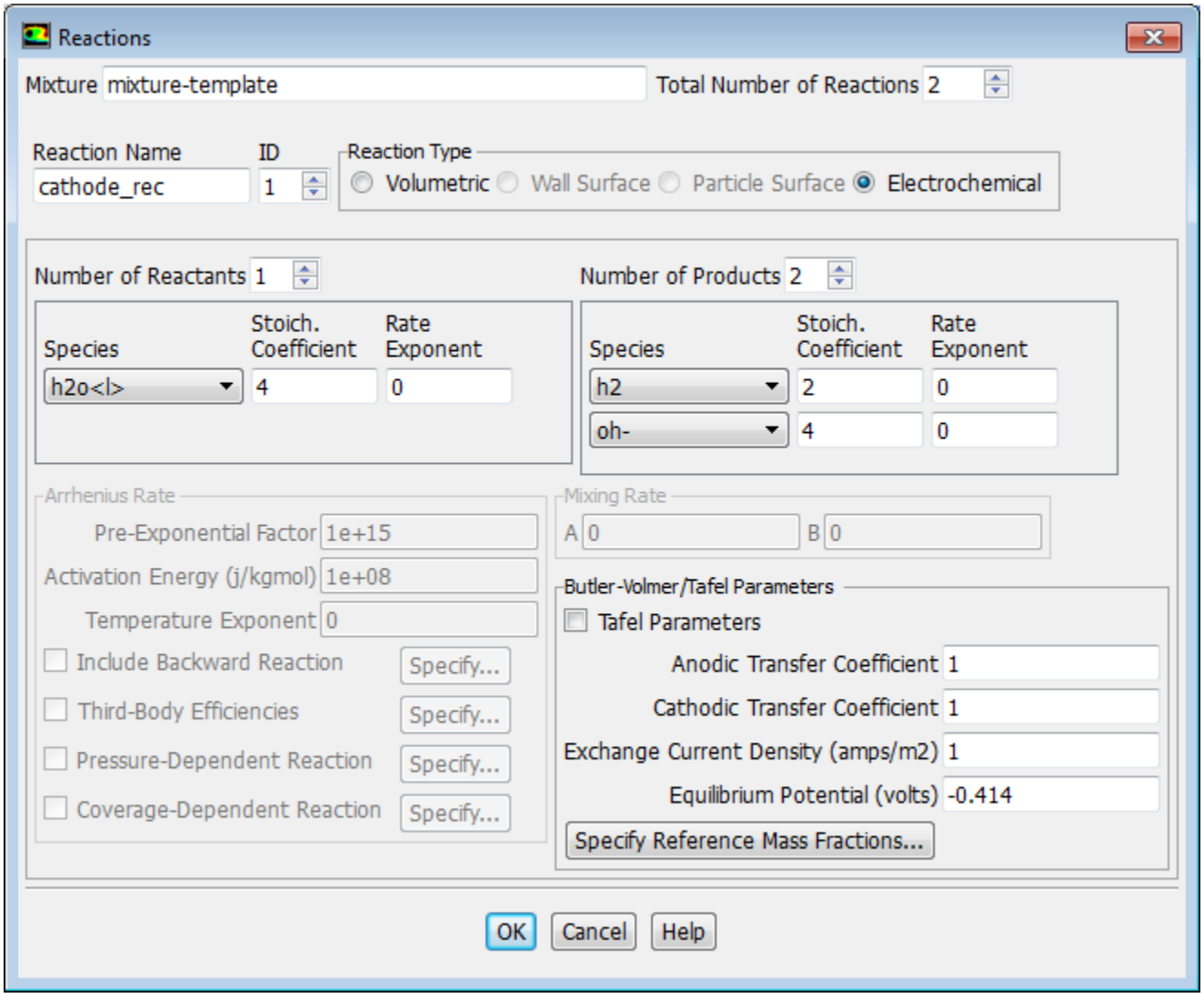
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. If solid species are involved in the electrochemical reactions (for example, in corrosion or electroplating simulations), add them to the **Selected Solid Species** list.

> **Note**
>
> Make sure that the bulk species is the last species in the **Selected Species** list. ANSYS Fluent solves transport equations for all but the last species in the **Selected Species** list. The order of the solution is the same as the order of the species in **Selected Species**.

5. Specify the reactions in the **Reactions** dialog box (opened by clicking **Edit** next to **Reactions** under **Properties** of the **Create/Edit Materials** dialog box).

Figure 14.18: The Reactions Dialog Box



For each reaction, configure the following settings:

a. Select **Electrochemical** as the reaction type.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. If you want the Tafel Equation 7.81 in the *Fluent Theory Guide*, rather than the Bulter-Volmer Equation 7.80, to be used in the kinetics calculation, select **Tafel Parameters**.

c. Specify the following **Butler-Volmer/Tafel Parameters**:

- Butler-Volmer parameters (the **Tafel Parameters** check box is cleared):
  - **Anodic Transfer Coefficient** (α_a in Equation 7.80 in the *Fluent Theory Guide*)
  - **Cathodic Transfer Coefficient** (α_c in Equation 7.80 in the *Fluent Theory Guide*)
- Tafel parameters (the **Tafel Parameters** check box is selected):
  - **Anodic Tafel Slope** ($\beta_{a,r}$ in Equation 7.81 in the *Fluent Theory Guide*)
  - **Cathodic Tafel Slope** ($\beta_{c,r}$ in Equation 7.81 in the *Fluent Theory Guide*)
- **Exchange Current Density** (i_o in Equation 7.80 in the *Fluent Theory Guide*)
- **Equilibrium Potential** (E_{eq} in Equation 7.82 in the *Fluent Theory Guide*)

d. Enter the k^{th} species dimensionless power (γ_k in Equation 7.80 in the *Fluent Theory Guide*) as **Rate Exponent**.

e. Under the **Butler-Volmer/Tafel Parameters** group box, click the **Specify Reference Mass Fractions...** button and in the **Reference Mass Fractions** dialog box that opens, enter the reference species mass fractions (Y_k in Equation 7.80 in the *Fluent Theory Guide*).

Note

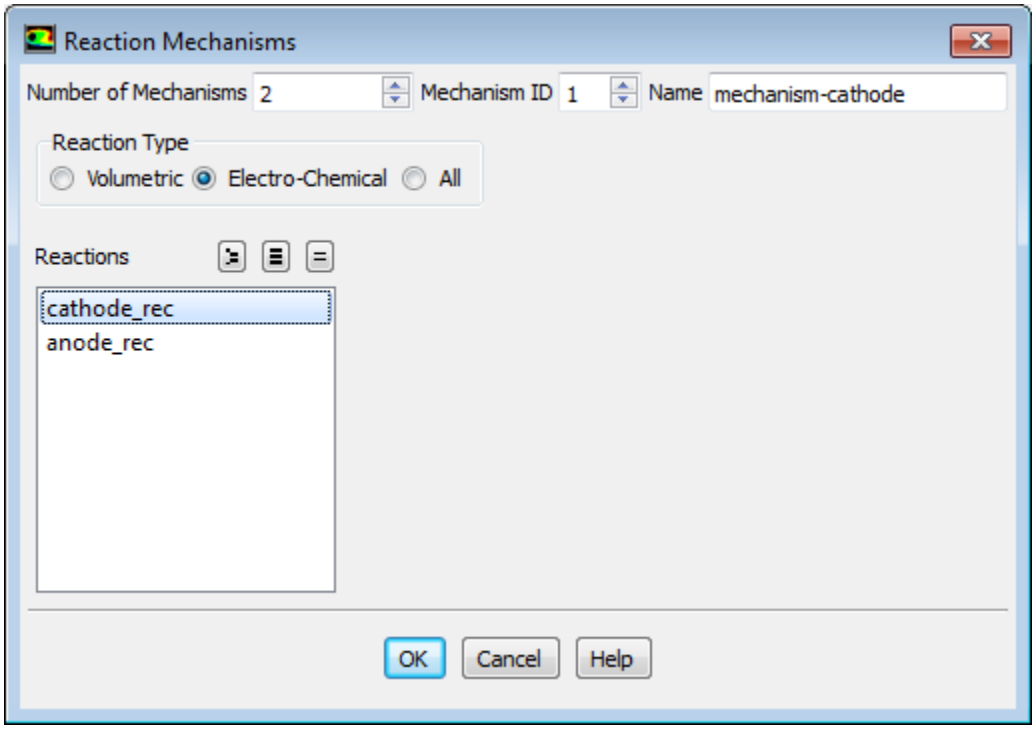
A user-defined function (UDF) hook for the electrochemical reaction rate is available as a `DEFINE_EC_RATE` function. Refer to `DEFINE_EC_RATE` in the *Fluent Customization Manual* for details about defining `DEFINE_EC_RATE` functions.

---

6. Define reaction mechanisms using the **Reaction Mechanisms** dialog box (opened by clicking **Edit** next to **Mechanism** under **Properties** of the **Create/Edit Materials** dialog box).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.19: The Reaction Mechanisms Dialog Box



a. Specify **Number of Mechanisms**. If you type in the value, be sure to press **Enter**.

b. For each electrochemical reaction mechanism, configure the following settings:

   i. Set the **Mechanism ID** of the mechanism you want to define. Again, if you type in the value, be sure to press **Enter**.

   ii. Select **Electrochemical** as a reaction type.

   iii. Under **Reactions**, select the reaction and in the **Name** text entry field, specify a meaningful name for the mechanism (for example, **`mechanism-anode`** or **`mechanism-cathode`**).

7. In the **Create/Edit Materials** dialog box, set the remaining mixture properties, such as **Density** (typically volume-weighted-mixing-law), **Mass Diffusivity**, and **Electrical Conductivity**.

   Note that if the anode, cathode, and electrolyte have different material properties, such as species mass diffusivities, it is best to use a UDF. Alternatively, you can create new materials for the anode and cathode by copying existing materials from the Fluent database, renaming them, and then assigning these materials to the anode and cathode fluid zones in the **Cell Zone Conditions** dialog box.

8. For electrolyte and electrode zones, set cell zone conditions using Cell Zone Conditions Task Page (p. 2367).

Setup → Cell Zone Conditions

a. In the **Fluid** dialog box, specify whether or not the selected zone is an electrolyte zone by selecting or clearing the **Electrolyte** check box. By default, all fluid zones are electrolyte zones.

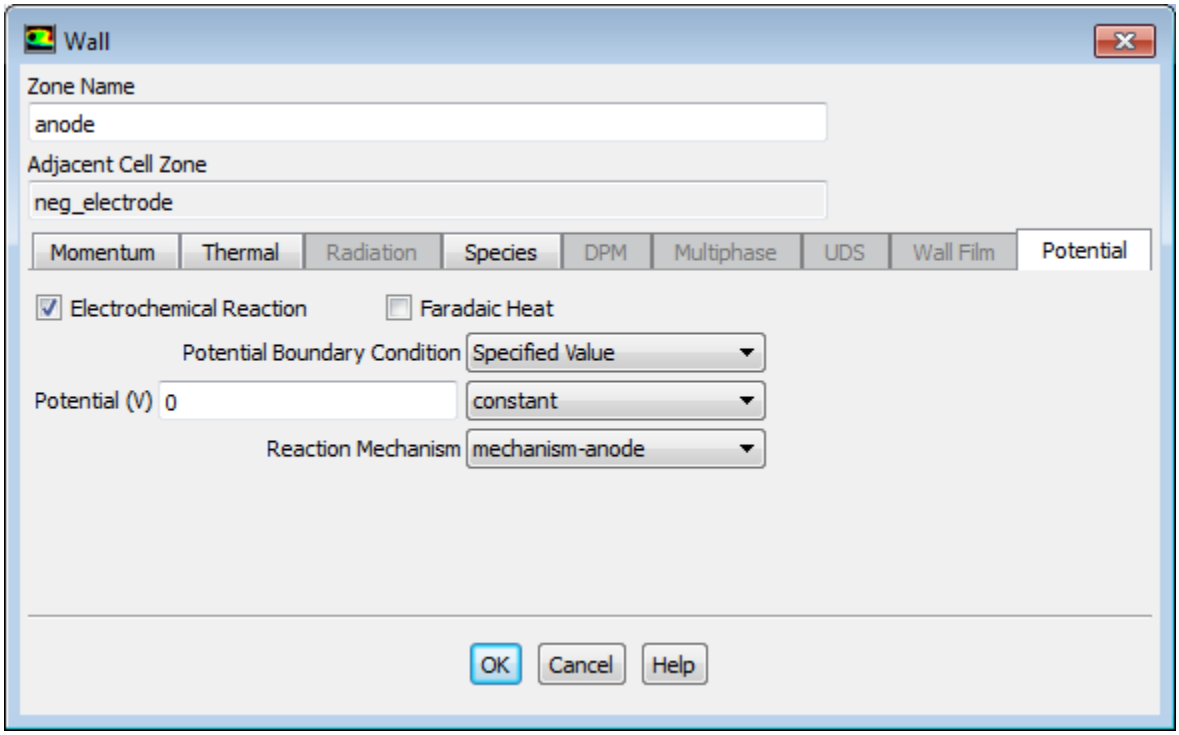
b. If there is no volumetric reactions, clear the **Reaction** check box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. Set the electrochemical boundary conditions for all relevant walls using Boundary Conditions Task Page (p. 2389).

Setup → Boundary Conditions

Figure 14.20: Wall Potential Boundary Condition



Electrochemical reaction can be defined at either exterior (1-sided) walls or interior (2-sided) walls with only one side exposed to electrolyte.

In the **Wall** dialog box, configure the following settings:

a. For exterior (1-sided) walls, set the potential boundary conditions as described in Using the Electric Potential Model (p. 1547).

b. Specify whether or not the wall is Faradaic by selecting or clearing the **Electrochemical Reaction** check box.

c. For Faradaic walls, from the **Reaction Mechanism** drop-down list, select the reaction mechanism that you have previously defined.

d. If the electrochemical reaction is defined on the wall (**Electrochemical Reaction** is selected), you can include the reaction heat in the energy equation by selecting the **Faradaic Heat** check box.

Note

For the 2-sided wall, you need to specify the potential boundary conditions only on one side of the wall. The settings you have specified will be automatically copied to the other side of the wall.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10. (optional) You can include additional sources in the energy, continuity, and species transport equations as described in the following sections:

    - Electrochemical Reaction Effects in the Energy Equation (p. 1010)
    - Electrochemical Reaction Effects in the Species Transport Equation (p. 1011)
    - Including Mass Transfer in Continuity (p. 1011).

11. Set the solution parameters in the Solution Methods Task Page (p. 2504) and Solution Controls Task Page (p. 2508).

    In many electrochemical applications, fluid convection can be neglected, in which case the **Flow** equation should be disabled in the **Equations** control box (accessible from the Solution Controls Task Page (p. 2508)).

    Similarly, for isothermal simulations, the **Energy** equation should be disabled.

    Important

    If the potential equation has difficulty converging, you can lower the **Faradaic Interface Current** under-relaxation factor in the **Solution Controls** task page to improve stability.

12. Once the solution is converged, you can display or report the following variables that are available under the **Potential** category:

    - **Electric Potential**
    - **Electric Conductivity**
    - **Faradaic Current Density**
    - **Total Surface Corrosion Rate**
    - **Electrode Surface Potential**
    - **Current Density**
    - **Joule Heat Source**
    - **Faradaic Heat Source**
    - **Total Echem Heat Source**

    In addition, you can display

    - **Echem Reaction Rate** of each reactions (under the **Reactions** category)
    - **Surface Corrosion Rate** of each solid species on a wall (under **Species** category)

14.4.3. Electrochemical Reaction Effects in the Energy Equation

When the electrochemical reaction model is enabled, you can specify two extra sources in the energy equation:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Joule heating in the fluid and solid zones

  You can include the Joule heating in the energy equation calculation by selecting the **Include Joule Heating in Energy Equation** option in the **Electric Potential** dialog box (Using the Electric Potential Model (p. 1547)).

- Electrochemical reaction heat at Faradaic interfaces

  You can include the electrochemical reaction heat by selecting the **Faradaic Heat** option in the boundary condition dialog boxes for each reacting wall.

14.4.4. Electrochemical Reaction Effects in the Species Transport Equation

When the electrochemical reaction model is enabled, you can specify an extra source in the species transport equation for the ion-species:

- Species migration in the electric field

You can include this effect by selecting the **Species Migration** option in the Species Model Dialog Box (p. 2216).

14.4.5. Including Mass Transfer in Continuity

When a solid species (corroded or deposited) is involved in an electrochemical reaction, the solid material can enter or leave the flow domain. Usually the mass transfer of the solid species is small and can be ignored in the continuity equation. However, if you want, you can include the mass source in the continuity equation by using the following Scheme command:

```
(rpsetvar 'potential/corrosion-source-in-continuity? #t)
```

14.4.6. Solution Procedures for Electrochemical Reactions

The strongly nonlinear coupling between species and electric field can cause convergence problems. To improve the stability of the solution, you can lower the **Faradaic Interface Current** under-relaxation factor in the Solution Controls Task Page (p. 2508).

In addition, you can try temporarily disabling the **Include Joule Heating in Energy Equation** and **Faradaic Heat** options (see Electrochemical Reaction Effects in the Energy Equation (p. 1010)).

You can enable these options once the solution becomes more stable.

14.5. Species Transport Without Reactions

In addition to the volumetric and surface reactions described in the previous sections, you can also use ANSYS Fluent to solve a species mixing problem without reactions. The species transport equations that ANSYS Fluent will solve are described in Volumetric Reactions in the Theory Guide, and the procedure you will follow to set up the non-reacting species transport problem is the same as that described in Overview of User Inputs for Modeling Species Transport and Reactions (p. 958) – Defining Other Sources of Chemical Species (p. 990), with some simplifications.

The basic steps are listed below:

1. Enable **Species Transport** in the **Species Model** dialog box and select the appropriate **Mixture Material**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → Models → Species Edit...

See Overview of User Inputs for Modeling Species Transport and Reactions (p. 958) for information about the mixture material concept, and Enabling Species Transport and Reactions and Choosing the Mixture Material (p. 960) for more details about using the **Species Model** dialog box.

2. (optional) If you want to model full multicomponent (Stefan-Maxwell) diffusion or thermal (Soret) diffusion, enable the **Full Multicomponent Diffusion** or **Thermal Diffusion** option.

3. Check and/or define the properties of the mixture and its constituent species.

   Setup → Materials

   Mixture properties include the following:

   - species in the mixture
   - other physical properties (for example, viscosity, specific heat)

   See Defining Properties for the Mixture and Its Constituent Species (p. 972) for details.

4. Set species boundary conditions, as described in Defining Cell Zone and Boundary Conditions for Species (p. 990).

No special solution procedures are usually required for a non-reacting species transport calculation. Upon completion of the calculation, you can display or report the following quantities:

- **Mass fraction of species-n**
- **Mole fraction of species-n**
- **Concentration of species-n**
- **Lam Diff Coef of species-n**
- **Eff Diff Coef of species-n**
- **Enthalpy of species-n** (pressure-based solver calculations only)
- **Relative Humidity** (when the water vapors (H_2O) is present in the species mixture material)
- **Mean Molecular Weight**
- **Liquid species mass fraction of species-n** (solidification and melting model only)

These variables are contained in the **Species...** and **Properties...** categories of the variable selection drop-down list that appears in postprocessing dialog boxes. See Field Function Definitions (p. 1931) for a complete list of flow variables, field functions, and their definitions. Displaying Graphics (p. 1747) and Reporting Alphanumeric Data (p. 1885) explain how to generate graphics displays and reports of data.

14.6. Reacting Channel Model

Information about using the reacting channel model is presented in the following subsections. For more information about the theoretical background, see Reacting Channel Model in the Theory Guide.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.6.1. Overview and Limitations of the Reacting Channel Model
14.6.2. Enabling the Reacting Channel Model
14.6.3. Boundary Conditions for Channel Walls
14.6.4. Postprocessing for Reacting Channel Model Calculations

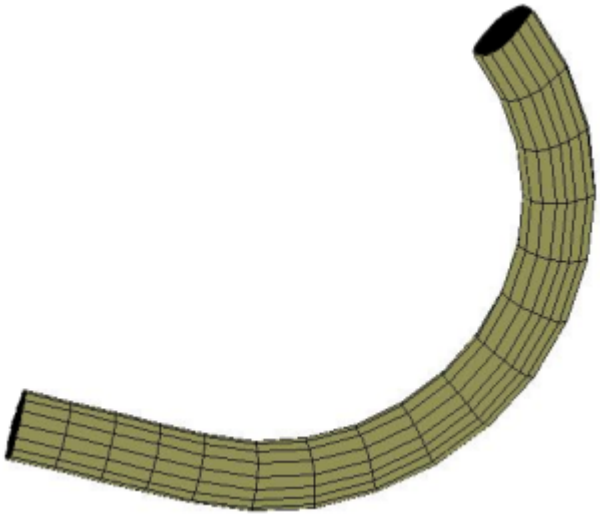
14.6.1. Overview and Limitations of the Reacting Channel Model

The reacting channel model in ANSYS Fluent offers an efficient solution methodology for solving reacting flow in shell and tube heat exchangers with long and thin channels. The volume inside the reacting channels is not meshed and solved with the outer flow. Instead, a plug flow approximation is used for the reacting channels, which is coupled to the outer flow through heat transfer across the channel walls.

The reacting channel model has the following limitations:

- The reacting channel model is available only for steady-state 3D problems.
- The reacting channel model solves the one-dimensional species equations at the centerline of the reacting channel. The accuracy of the model depends on how well ANSYS Fluent computes the channel centerline from the channel wall surface mesh. It is highly recommended to use a swept structured surface mesh on the channel wall (see Figure 14.21: Optimal Surface Mesh on the Reacting Channel Wall (p. 1013)). Unstructured channel wall surface meshes may result in loss of accuracy due to inconsistent channel centerlines.

Figure 14.21: Optimal Surface Mesh on the Reacting Channel Wall



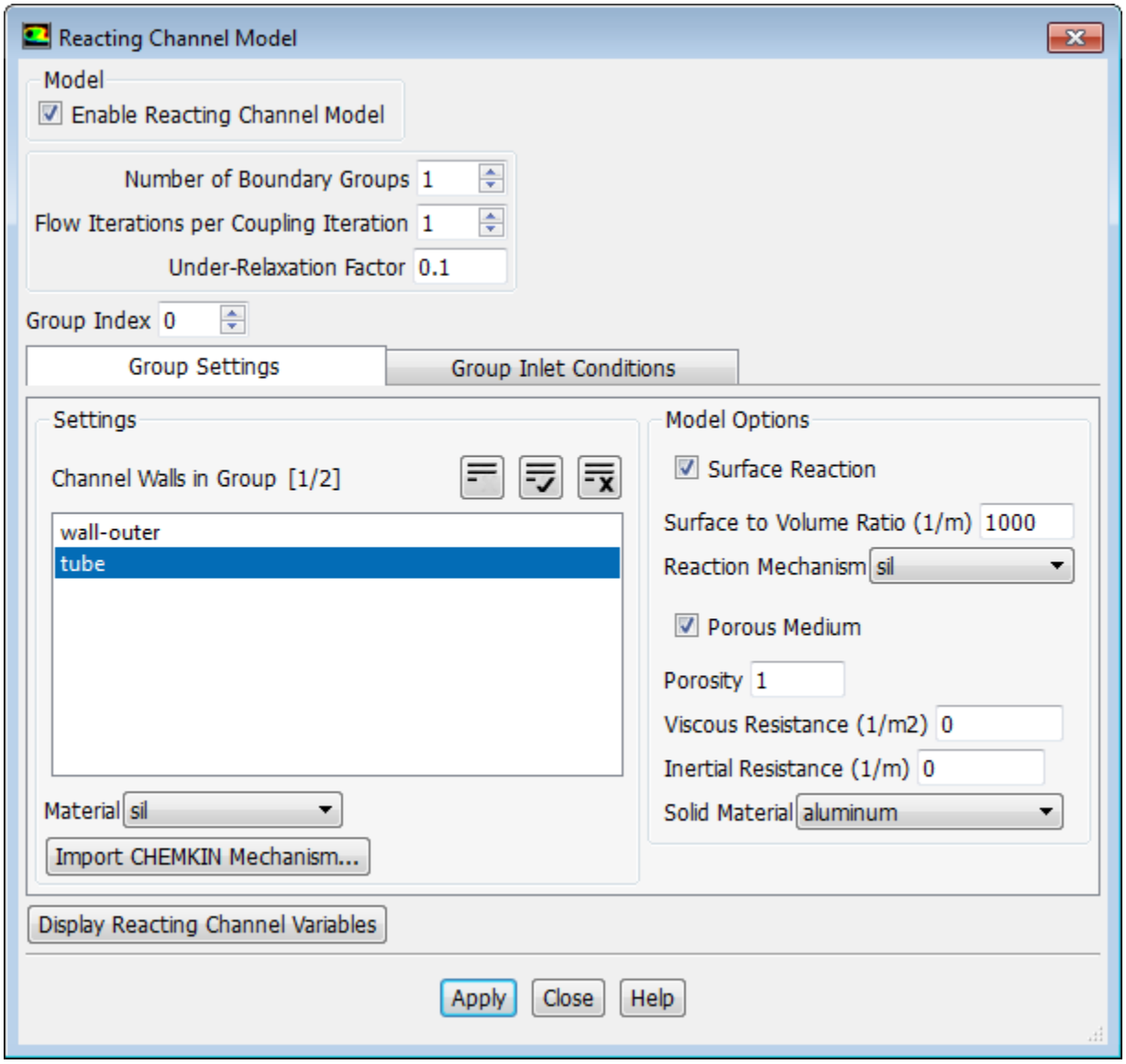
14.6.2. Enabling the Reacting Channel Model

The steps and procedure to set up a reacting channel model are outlined below. In this section, the steps pertinent to the reacting channel model only are explained. The setting up of other models used in conjunction with reacting channel models are explained in other sections of the User's Guide.

1. Enable the reacting channel model.

Setup → Models → Reacting Channel Model → Edit...

Figure 14.22: The Reacting Channel Model Dialog Box



When you enable the **Enable Reacting Channel Model** option, the dialog box will expand to show the relevant inputs (see Figure 14.22: The Reacting Channel Model Dialog Box (p. 1014)).

2. Specify the **Number of Boundary Groups**. If you have multiple channels, you can group together channels with common flow direction, mixture materials, inlet compositions, temperature, pressure, and mass flow rate. You will be required to provide the inputs for each boundary group.

> **Note**
>
> If you have multiple tubes, you will need to define each tube as a different wall and set up the model by grouping the walls together if they have the same boundary conditions.

3. Enter the number of **Flow Iterations Per Coupling Iteration**. This is the number of outer flow iterations for each channel flow iteration.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Optionally change an **Under-Relaxation Factor**. The value of the under-relaxation factor is used to update the heat flux from the reacting channel. See Equation 7.107 in the Theory Guide.

5. Set the **Group Index**. Note that the **Group Index** for the first boundary group is set to `0`.

   The inputs for each boundary group are entered in the **Group Settings** and **Group Inlet Conditions** tabs as further described.

6. Under the **Group Settings** tab, specify the model settings and options. (see Figure 14.23: The Reacting Channel Model Dialog Box (Group Inlet Conditions Tab) (p. 1016)).

   a. In the **Channel Walls in Group** selection list, select the wall boundary zones that correspond to the reacting channels.

   b. Select the group material from the **Material** drop-down list. You can choose any mixture material available in the case as a group material.

   c. Optionally, import a CHEMKIN mechanism using **Import CHEMKIN Mechanism...** button. For details see Importing a Volumetric Kinetic Mechanism in CHEMKIN Format (p. 967).

   d. To model surface reactions within the channel, enable **Surface Reactions** under **Model Options** group box. The dialog box will expand to show the related inputs as shown in Figure 14.22: The Reacting Channel Model Dialog Box (p. 1014). These inputs include:

      - **Surface to Volume Ratio** of the channel group
      - Surface reaction mechanism of the group (selected from the **Reaction Mechanism** drop-down list)

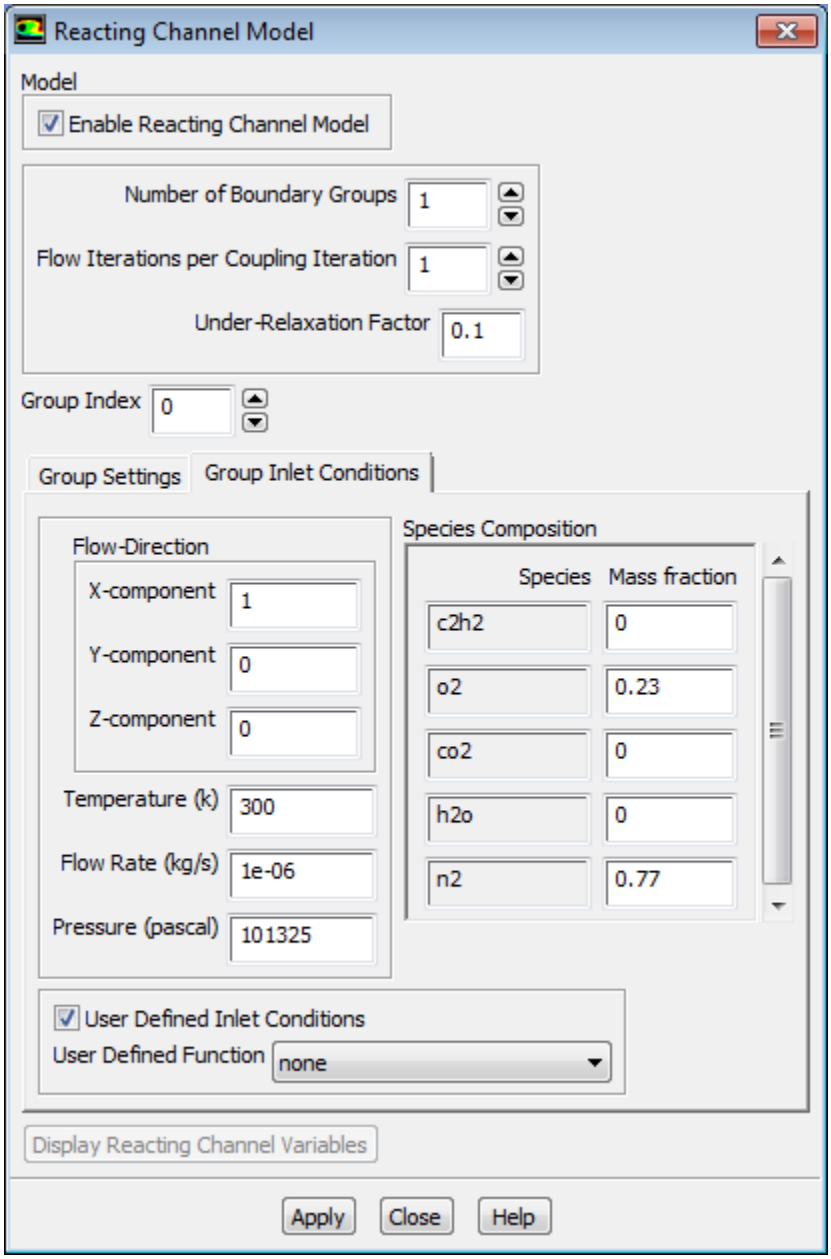
   e. To model porous medium, enable **Porous Medium** under **Model Options** group box. The dialog box will expand to show porous medium inputs as shown in Figure 14.22: The Reacting Channel Model Dialog Box (p. 1014). These inputs include:

      - **Porosity** of the channel
      - **Viscous Resistance** in axial direction of the channel
      - **Inertial Resistance** in axial direction of the channel
      - Solid material of the porous medium inside the channel (selected from the **Solid Material** drop-down list)

7. Under the **Group Inlet Conditions** tab, specify the inlet conditions for the boundary group (see Figure 14.23: The Reacting Channel Model Dialog Box (Group Inlet Conditions Tab) (p. 1016)). You can specify the inlet condition parameters as constant values. ANSYS Fluent provides you also with the option of redefining any or all of the inlet condition parameters using user-defined functions (UDFs).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.23: The Reacting Channel Model Dialog Box (Group Inlet Conditions Tab)



a. Specify the **X-**, **Y-**, and **Z-component** of the flow at the inlets of the channels in the current group in the **Flow Direction** group box.

Note

Since there is no surface or zone to identify the channel inlet or outlet, the inlet flow direction of the group is used to determine the channel inlet. Therefore, It is very important to specify the flow direction correctly.

---

b. Enter the inlet channel **Temperature**, **Flow Rate** and **Pressure** in the group.

c. Specify the inlet mass fractions for each species in the group in the **Species Composition** group box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

d. To specify an inlet condition using a user-defined function (UDF), select the **User Defined Inlet Conditions** check box and then select the appropriate user-defined function from the drop-down list. For information about specifying and hooking DEFINE_REACTING_CHANNEL_BC UDFs, see `DEFINE_REACTING_CHANNEL_BC` in the *Fluent Customization Manual*.

8. Click the **Apply** button to save the settings.

9. If you specified more than one boundary group in step 1, change the group ID value in **Group Index** integer number entry box and repeat steps 5–7 for each remaining boundary group.

Once the reacting channel model is defined, you can generate reports or plots of reacting channel variables using the **Reacting channel 2D Curves** dialog box (Figure 14.25: Reacting Channel 2D Curves Dialog Box (Plot) (p. 1019)). You can open the **Reacting channel 2D Curves** dialog box by clicking the **Display Reacting Channel Variables** button in the **Reacting Channel Model** dialog box. For information about the plotting of reacting channel model variables, see Postprocessing for Reacting Channel Model Calculations (p. 1018).

Modeling Curvilinear Reacting Channels

In the reacting channel model, the two ends of a channel are identified as the inlet and the outlet based on the inputs provided for the inlet flow direction of the group. However, for some special channel configurations, such as U-tubes, it is not possible to identify the inlet/outlet of the channel based on the information of the inlet flow direction. For such cases, an additional input is required to differentiate between the end points of the channel as inlet or outlet. The additional input is to provide the centroid coordinates of the inlet of the U shaped channel. Therefore, for U-tube configurations, after setting up the reacting channel model following the earlier steps in this section, it is required to use the following text user interface commands to set the inlet and outlets correctly:

```
/define/models/species> reacting-channel-model-options
Are any of the channels a U-tube configuration? [no] yes
Is wall tube1 a U-tube configuration? [no] yes
Enter the coordinates of the center of the inlet for wall tube1
X coordinate [0] 0.01
Y coordinate [0] 0
Z coordinate [0] 0
```

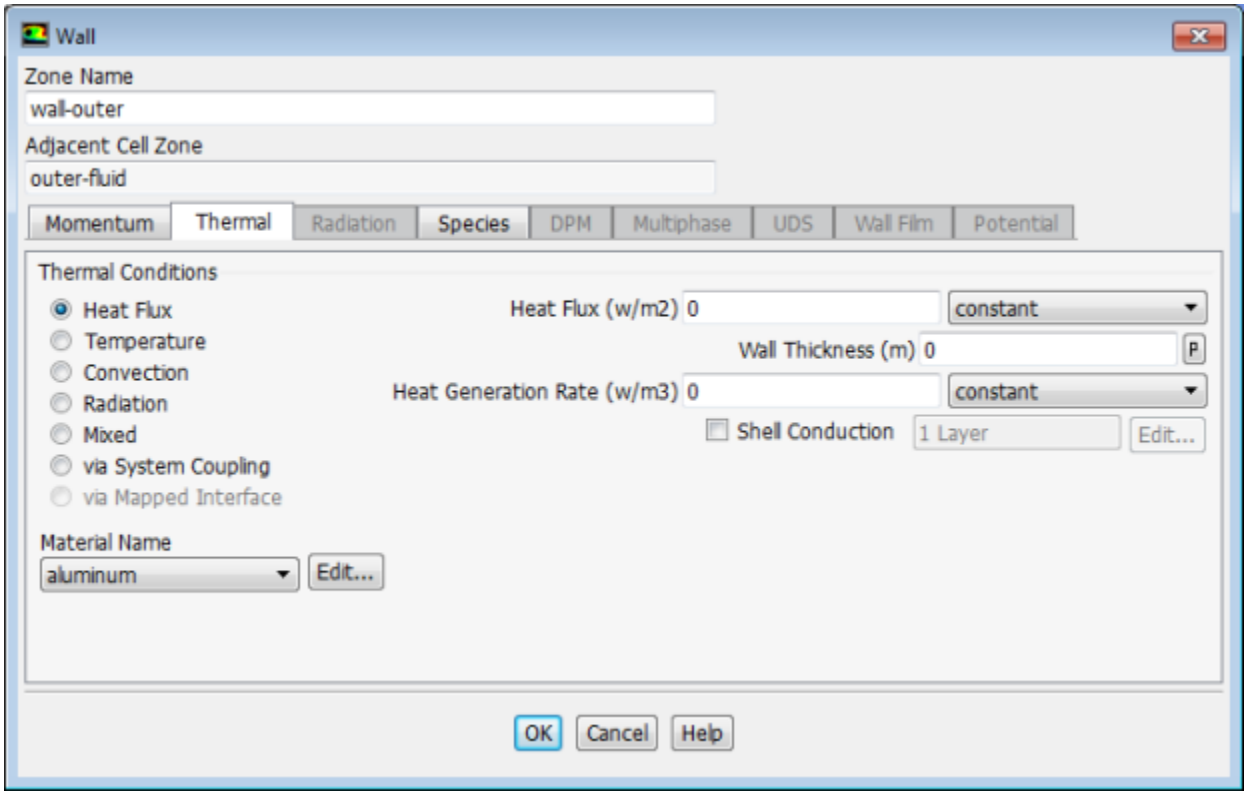
Where `tube1` is a reacting channel wall having a U shape. You will be asked the same questions for each of the existing reacting channel walls.

14.6.3. Boundary Conditions for Channel Walls

In the ANSYS Fluent wall boundary conditions dialog box, the wall surfaces that you selected as channel walls in the **Reacting Channel Model** dialog box will have zero heat flux boundary conditions, which are disabled as shown in Wall Surface Reactions and Chemical Vapor Deposition (p. 995)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.24: The Wall Boundary Condition Dialog Box for the Reacting Channel Model



The heat flux at the channel walls is calculated internally through the coupling of the reacting channel model with the outer flow as described in Reacting Channel Model in the Theory Guide. For all other wall surfaces, which are not channel walls, all boundary conditions as described in Wall Boundary Conditions (p. 358) are applicable.

14.6.4. Postprocessing for Reacting Channel Model Calculations

ANSYS Fluent provides postprocessing options for plotting and reporting channel variables using the **Reacting Channel 2D Curves** dialog box. You can generate X-Y plots or reports of the following reacting channel variables:

- Bulk mean temperature of the channel
- Wall temperature of the channel
- Nusselt Number (for the channel wall on the channel side)
- Wall heat flux through the channel walls
- Velocity inside the channel
- Density of the channel
- Mass fraction of the species inside the channel
- The surface coverage and the deposition rate of the surface species.

The steps for generating the plots/reports are as follows:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select either **Plot Reacting Channel Variables** or **Report Reacting Channel Outlet Average**.

   The **Reacting Channel 2D Curves** dialog box displays either the **Variable Name** drop-down list for selecting a single species for X-Y plot generation, or the **Variable Names** selection list for selecting multiple species for report generation as shown in Figure 14.25: Reacting Channel 2D Curves Dialog Box (Plot) (p. 1019) and Figure 14.26: Reacting Channel 2D Curves Dialog Box (Report) (p. 1020).

Figure 14.25: Reacting Channel 2D Curves Dialog Box (Plot)

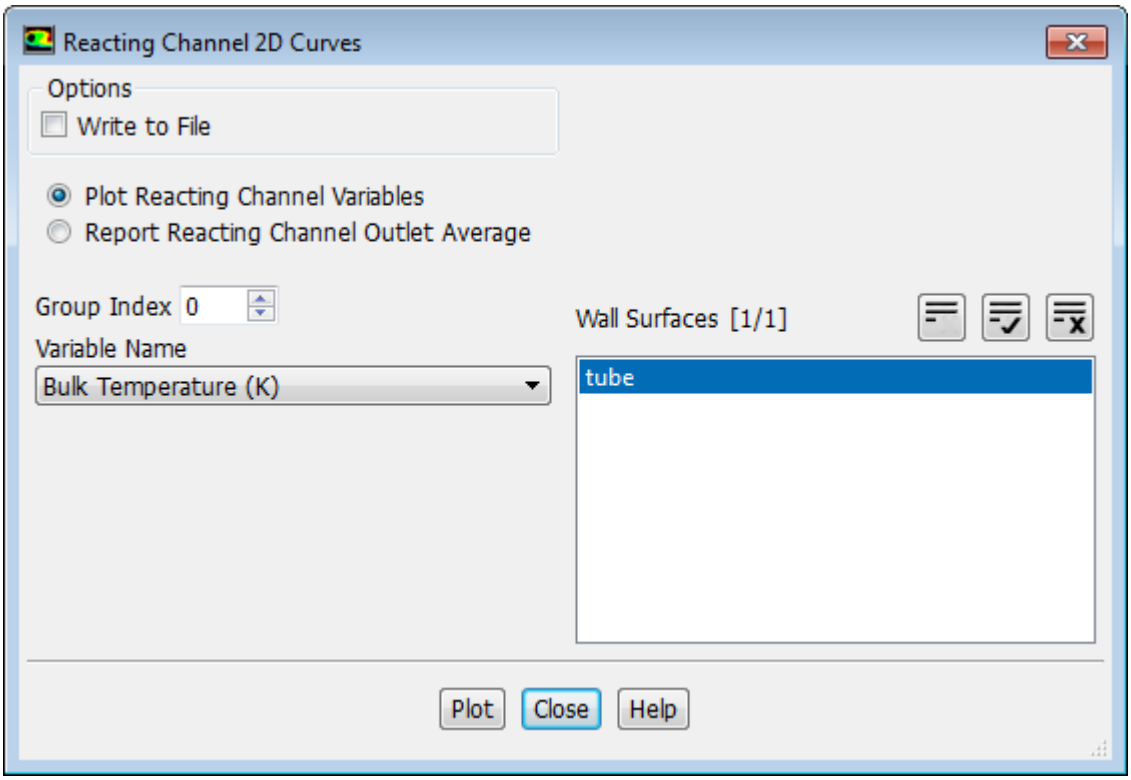
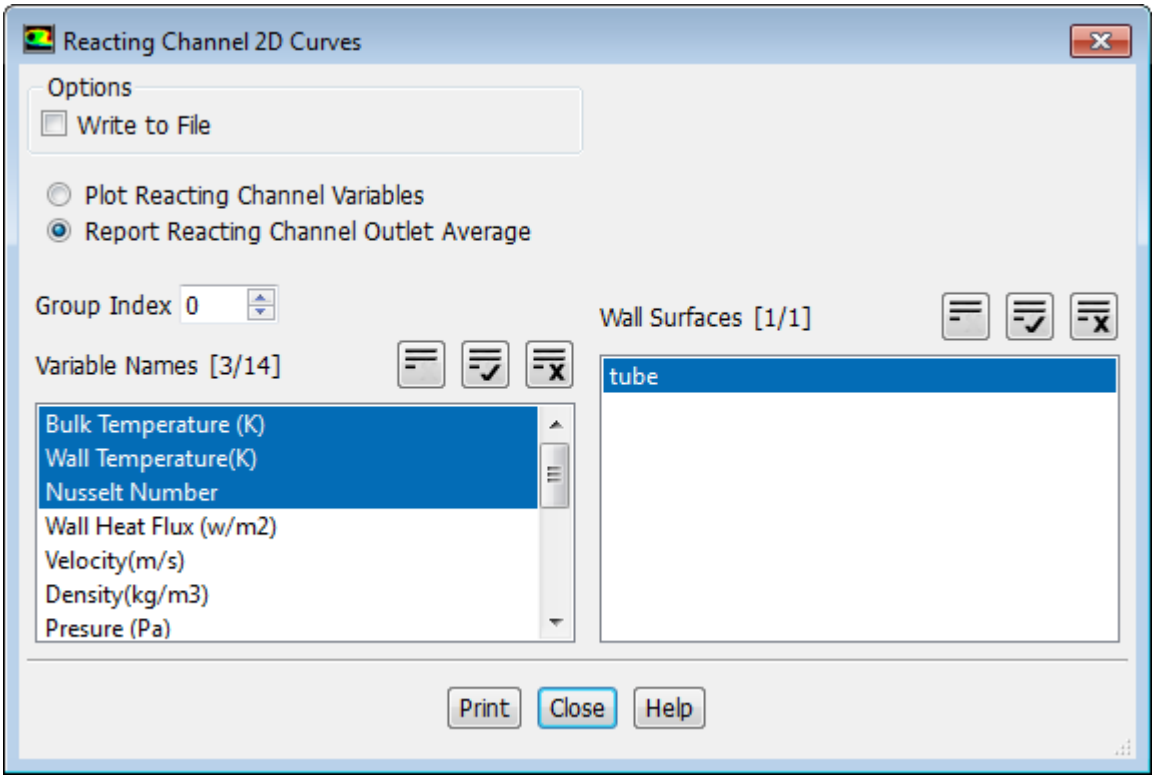


Figure 14.26: Reacting Channel 2D Curves Dialog Box (Report)



2. Specify whether you want to write the plot data to a file by using the **Write To File** check box.

3. If more than one boundary group is defined for reacting channel model, specify the group ID in the **Group Index** integer number entry box.

4. The **Variable Name** drop-down/**Variable Names** selection list contains reacting channel variables of the specified boundary group. Select the reacting channel variable(s) from the list.

5. Select the wall surface(s) from the **Wall Surfaces** selection list on which the plot/report will be generated (at least one wall surface must be selected).

6. Click one of the following as applicable:

   - **Plot** to view X-Y plot in the graphics window
   - **Print** to view report in the console window
   - **Write...** to save to file

14.7. Reactor Network Model

Information about using the Reactor Network model is presented in the following subsections.

14.7.1. Overview and Limitations of the Reactor Network Model
14.7.2. Solving Reactor Networks
14.7.3. Postprocessing Reactor Network Calculations

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For more information about the theoretical background, see Reactor Network Model in the *Fluent Theory Guide*.

14.7.1. Overview and Limitations of the Reactor Network Model

The Reactor Network model allows rapid solution of combustors using detailed chemical kinetic mechanisms. This speed is achieved by first performing a Fluent CFD simulation with a fast chemistry model, such as a Non-Premixed, Partially-Premixed, or Eddy-Dissipation model, and then using the Reactor Network model to solve with a detailed kinetic mechanism. The Reactor Network model agglomerates cells from the CFD solution into a user-specified small number of reactors and computes detailed chemistry on this reactor network.

Since the chemical kinetics is decoupled from the flow, reactor network simulations are applicable to steady simulations where the chemical kinetics does not affect the flow significantly, such as pollutant formation. Inherently unsteady simulations, such as ignition and global extinction, cannot be modeled with reactor networks. Note that the Reactor Network model can be used for unsteady simulations of statistically steady flows, such as LES, where the time-averaged flow and species fields are used to construct the reactor network.

14.7.2. Solving Reactor Networks

The steps and procedure for setting up and solving a Reactor Network model are outlined below.

1. Enable the Reactor Network model.

   Setup → **Models** → **Species** → **Reactor Network** **Edit...**

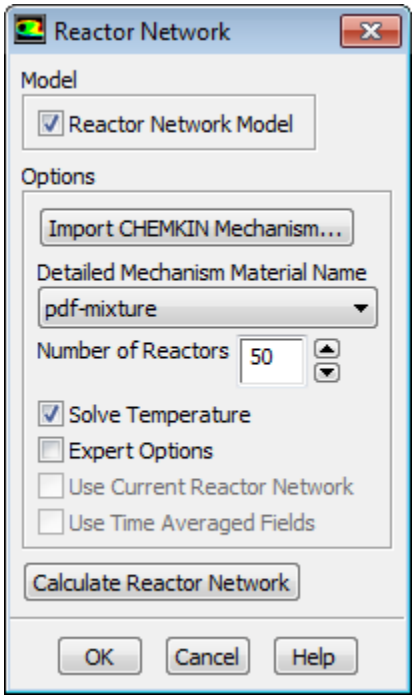
   Important

   Note that this model is only available when one of the reacting species models is enabled.

   When you select the **Reactor Network Model** option, the dialog box expands to show the relevant modeling controls (see Figure 14.27: Reactor Network Dialog Box (Steady-State Flow) (p. 1022)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.27: Reactor Network Dialog Box (Steady-State Flow)



2. If a detailed chemical mechanism has already been imported, select the mixture material from the **Detailed Mechanism Material Name** drop-down list. Otherwise import a mechanism in CHEMKIN format by clicking **Import CHEMKIN Mechanism...**. The mechanism files are imported into your case using the **CHEMKIN Mechanism Import** dialog box as described in Specifying a Chemical Mechanism File for Flamelet Generation (p. 1037).

3. Set the **Number of Reactors** in the reactor network. The default value of 50 is a good starting value, but it can be increased for greater accuracy.

4. Specify whether or not the temperature is calculated from the equation of state by selecting or clearing the **Solve Temperature** check box. By default, this option is selected, and ANSYS Fluent will determine the temperatures in the reactor network.

5. Once cells are agglomerated and a reactor-network solution is calculated, you can select the **Use Current Reactor Network** option to perform further reactor network simulations with the existing network of connected reactors. This option can be used to continue iterations from the previous reactor- network solution if additional convergence is desired. The option also allows you to change the chemical mechanism or species boundary conditions to generate a new solution on the current reactor network.

6. (for transient analysis with the **Data Sampling For Time Statistics** option selected in the **Run Calculation** task page only) Specify whether or not you want to use the time-averaged composition fields to agglomerate the reactors and time-averaged velocity fields to calculate the reactor mass flux matrix (instead of the instantaneous fields) by selecting or clearing the **Use Time Averaged Fields** option.

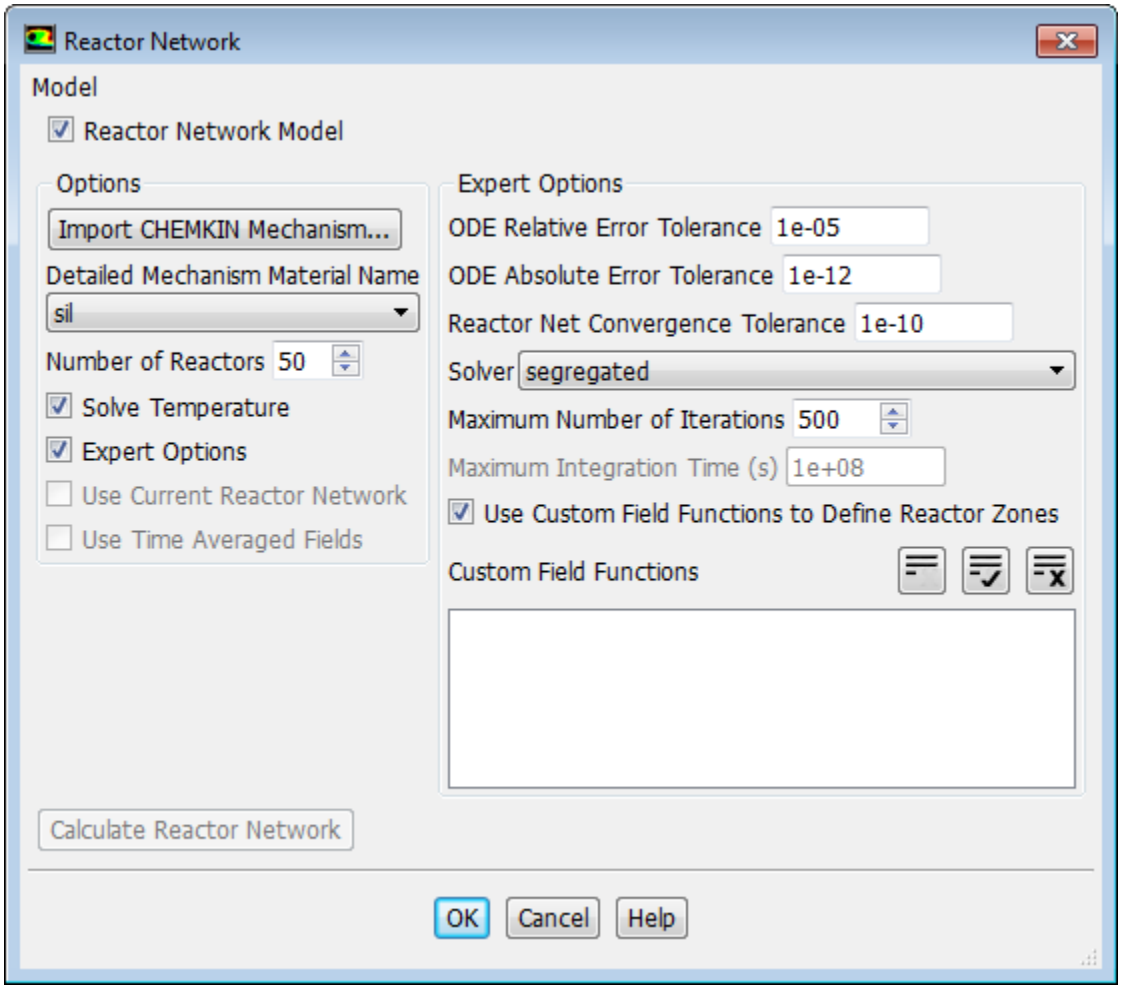
Note

The **Use Time Averaged Fields** option should be used for unsteady simulations (for example, LES) of statistically stationary combustors.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. If necessary, specify custom-field functions and adjust the default solver settings using the **Expert Options** control. When **Expert Options** is selected, the **Reactor Network** dialog box expands to reveal the following advanced options:

Figure 14.28: Reactor Network Dialog Box - Expert Options



ODE Relative Error Tolerance, ODE Absolute Error Tolerance

The default values for the ODE relative and absolute error tolerances are 1e-05 and 1e-12, respectively. If the ODE solver fails to converge, it is often helpful to lower the **ODE Relative Error Tolerance** and the **ODE Absolute Error Tolerance**.

Reactor Network Convergence Tolerance

The default value for the reactor network convergence tolerance is 1e-08. Residuals should be well converged at the default tolerance. Higher tolerances may be acceptable for engineering accuracy.

Solver

The default segregated solver typically converges faster than the coupled solver. However, when using the segregated solver, residuals may stall above an acceptable tolerance. In this case, the coupled solver should be used.

Maximum Number of Iterations (segregated solver)

If residuals are still decreasing and better convergence is desired, increase the maximum number of iterations. If residuals are stalled and a quicker exit from the solver is desired, decrease the maximum number of iterations.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Maximum Integration Time (coupled solver)
The ODE solver terminates the calculation at this time if residuals fail to converge.

Use Custom Field Functions to Define Reactor Zones
When selected, this option allows you to use custom field functions (Custom Field Functions (p. 2003)) as composition variables to create the reactor volumes. ANSYS Fluent clusters CFD cells that have similar compositions using the following default composition variables:

- temperature and mixture fraction for Non-Premixed and Partially-Premixed cases
- temperature and mass fractions of N_2 and H_2O for Species Transport cases

You can override these default composition variables by selecting **Use Custom Field Functions to Define Reactor Zones** and then selecting up to four custom field functions from the **Custom Field Functions** selection list. ANSYS Fluent will group CFD cells that are close in these custom field function composition variables. In the example shown in Figure 14.28: Reactor Network Dialog Box - Expert Options (p. 1023), cells with similar turbulent kinetic energy, x coordinate, and y coordinate will be agglomerated to form the reactor network.

8. Click **Calculate Reactor Network** to obtain a flow solution.

   As the calculation is progressing, ANSYS Fluent prints the reactor network residuals for each iteration in the console.

   Note

   The reactor network solution should converge with the default settings. However, if the model fails to converge with sufficient accuracy, you can adjust the default reactor network solver settings by using the **Expert Options** control.

14.7.3. Postprocessing Reactor Network Calculations

ANSYS Fluent provides additional postprocessing options for reactor network calculations. The following reactor network variables are available for generating graphical plots:

- **Reactor Net Zone ID**
- **Reactor Net Temperature**
- **Reactor Net Mass fraction of species-n**

These variables will appear under the **Reactor Network...** postprocessing category of the variable selection drop-down lists.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 15: Modeling Non-Premixed Combustion

In non-premixed combustion, fuel and oxidizer enter the reaction zone in distinct streams. This is in contrast to premixed systems, in which reactants are mixed at the molecular level before burning. Examples of non-premixed combustion include pulverized coal furnaces, diesel internal-combustion engines and pool fires.

Under certain assumptions, the thermochemistry can be reduced to a single parameter: the mixture fraction. The mixture fraction, denoted by f, is the atomic mass fraction that originated from the fuel stream. That is, it is the local mass fraction of burnt and unburnt fuel stream elements (C, H, and so on) in all the species (CO_2, H_2O, O_2, and so on). The approach is elegant because atomic elements are conserved in chemical reactions. In turn, the mixture fraction is a conserved scalar quantity, and therefore its governing transport equation does not have a source term. Combustion is simplified to a mixing problem, and the difficulties associated with closing nonlinear mean reaction rates are avoided. Once mixed, the chemistry can be modeled as being in chemical equilibrium with the **Chemical Equilibrium** model, being near chemical equilibrium with the **Steady Diffusion Flamelet** model, or significantly departing from chemical equilibrium with the **Unsteady Diffusion Flamelet** model.

The non-premixed combustion model is presented in the following sections:

15.1. Steps in Using the Non-Premixed Model
15.2. Setting Up the Equilibrium Chemistry Model
15.3. Setting Up the Steady and Unsteady Diffusion Flamelet Models
15.4. Defining the Stream Compositions
15.5. Setting Up Control Parameters
15.6. Calculating the Flamelets
15.7. Calculating the Look-Up Tables
15.8. Standard Files for Diffusion Flamelet Modeling
15.9. Setting Up the Inert Model
15.10. Defining Non-Premixed Boundary Conditions
15.11. Defining Non-Premixed Physical Properties
15.12. Solution Strategies for Non-Premixed Modeling
15.13. Postprocessing the Non-Premixed Model Results

For theoretical background on the non-premixed combustion model, see Non-Premixed Combustion in the Theory Guide.

15.1. Steps in Using the Non-Premixed Model

A description of the inputs for the non-premixed model is provided in the sections that follow.

15.1.1. Preliminaries
15.1.2. Defining the Problem Type
15.1.3. Overview of the Problem Setup Procedure

15.1.1. Preliminaries

Before turning on the non-premixed combustion model, you must enable turbulence calculations in the Viscous Model Dialog Box (p. 2176).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → Models → Viscous Edit...

If your model is non-adiabatic, you should also enable heat transfer (and radiation, if required).

Setup → Models → Energy ON

Setup → Models → Radiation Edit...

Figure 8.7: Reacting Systems Requiring Non-Adiabatic Non-Premixed Model Approach in the Theory Guide illustrates the types of problems that must be treated as non-adiabatic.

15.1.2. Defining the Problem Type

Your first task is to define the type of reaction system and reaction model that you intend to use. This includes selection of the following options:

- Non-premixed or partially premixed model option (see Modeling Partially Premixed Combustion (p. 1099)).
- Equilibrium chemistry model, steady diffusion flamelet model, unsteady diffusion flamelet model, or diesel unsteady flamelet.
- Adiabatic or non-adiabatic modeling options (see Non-Adiabatic Extensions of the Non-Premixed Model in the Theory Guide).
- Addition of a secondary stream (equilibrium model only).
- Empirically defined fuel and/or secondary stream composition (equilibrium model only).

You can make these model selections using the **Species Model** dialog box (Figure 15.6: The Species Model Dialog Box (Chemistry Tab) (p. 1032)).

Setup → Models → Species Edit...

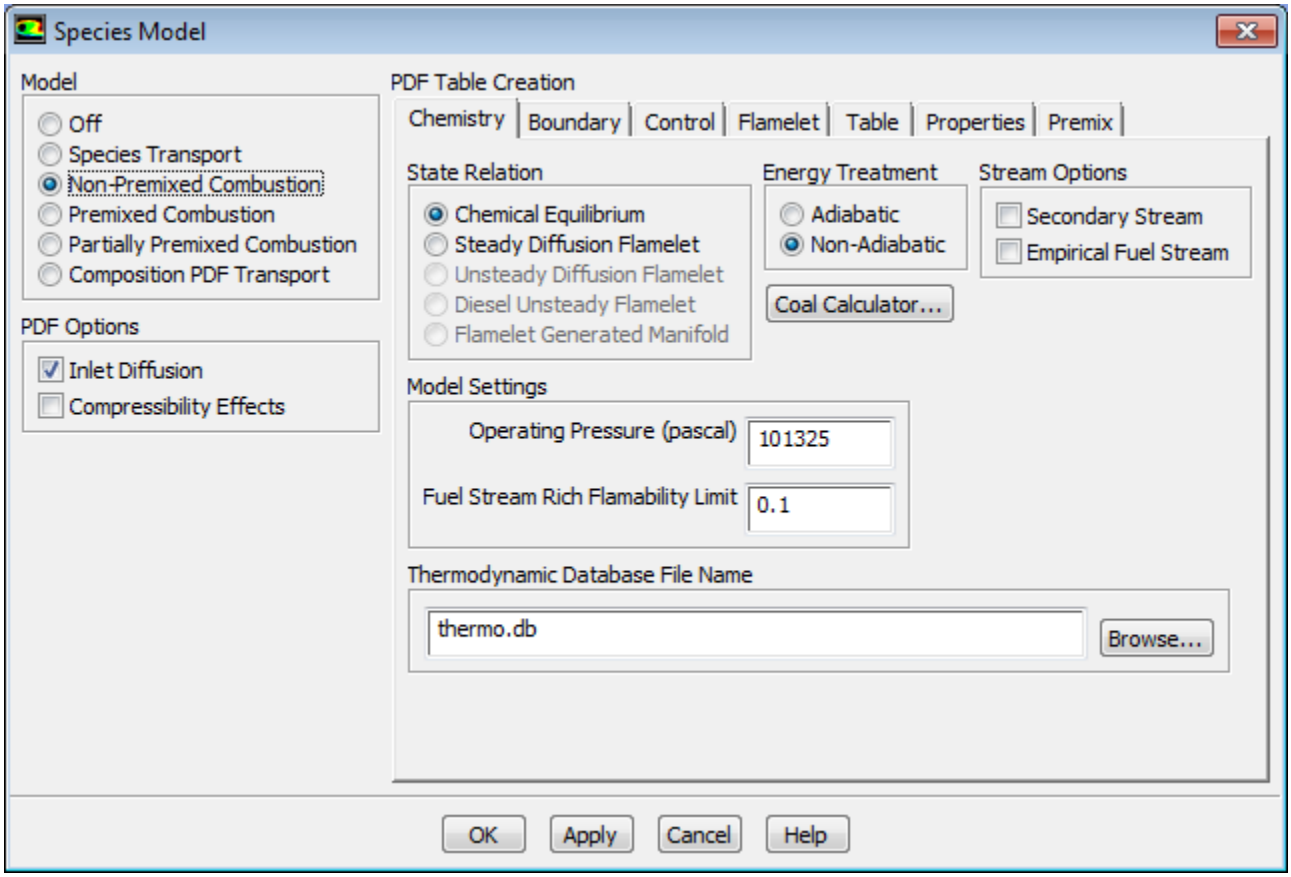
15.1.3. Overview of the Problem Setup Procedure

For a single-mixture-fraction problem, you will perform the following steps:

1. Choose the chemical description of the system: chemical equilibrium, steady diffusion flamelet, unsteady diffusion flamelet, or diesel unsteady flamelet (Figure 15.1: Defining Equilibrium Chemistry (p. 1027)).
2. Indicate whether the problem is adiabatic or non-adiabatic.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

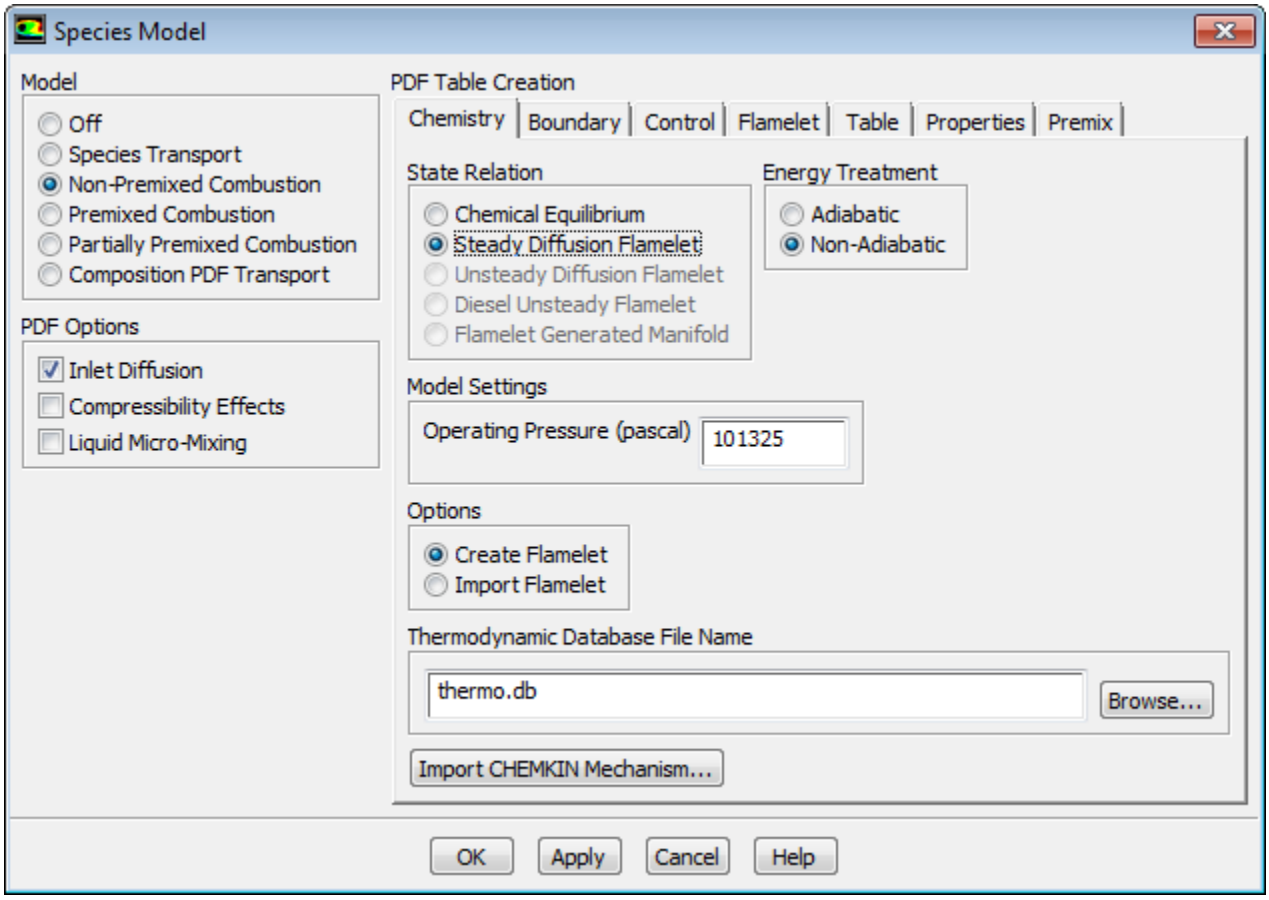
Figure 15.1: Defining Equilibrium Chemistry



3. (steady diffusion flamelet model only) Import a flamelet file or appropriate CHEMKIN mechanism file if generating flamelets (Figure 15.2: Defining Steady Diffusion Flamelet Chemistry (p. 1028)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

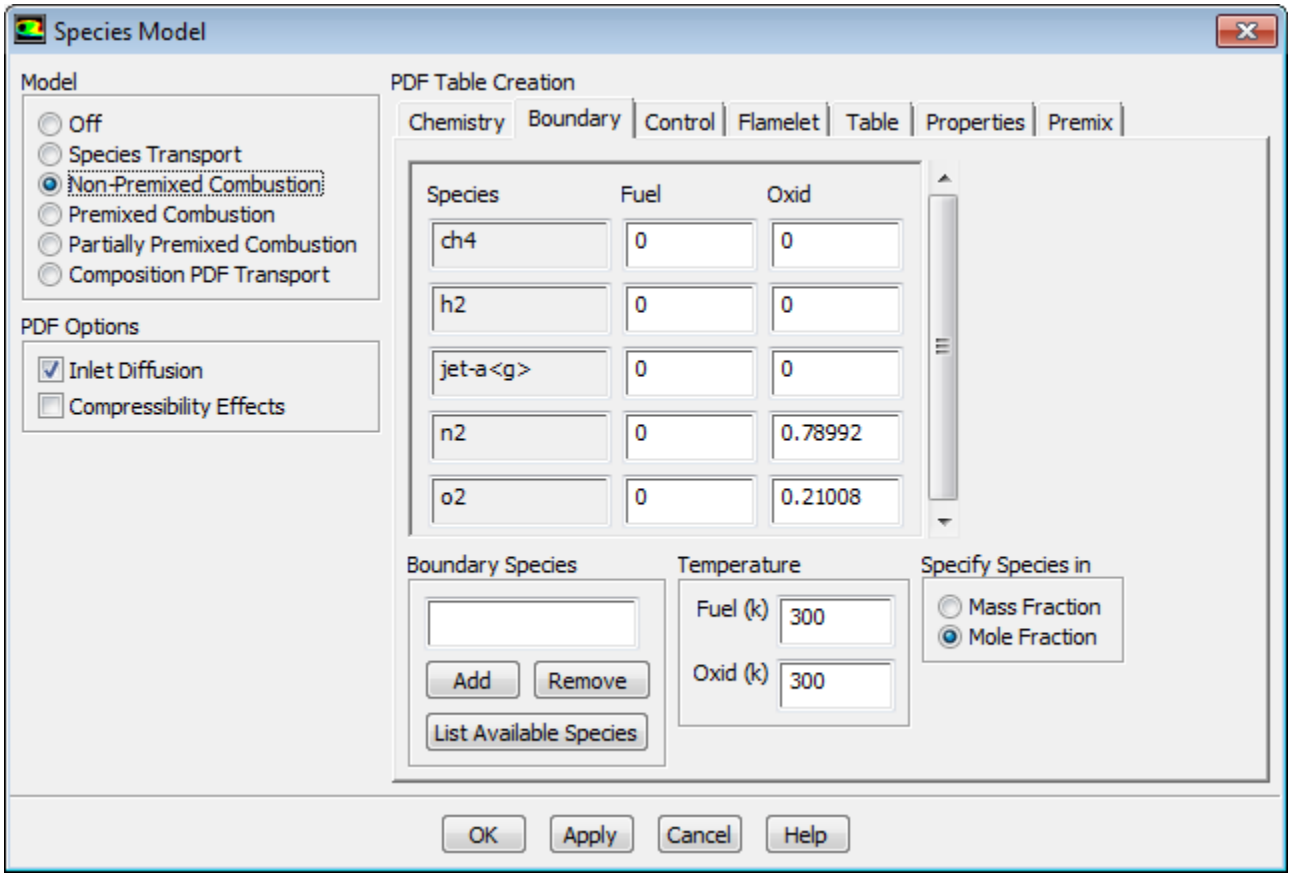
Figure 15.2: Defining Steady Diffusion Flamelet Chemistry



4. Define the chemical boundary species to be considered for the streams in the reacting system model. Note that this step is not relevant in the case of flamelet import (Figure 15.3: Defining Chemical Boundary Species (p. 1029)). For more information, see Defining the Stream Compositions (p. 1042).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

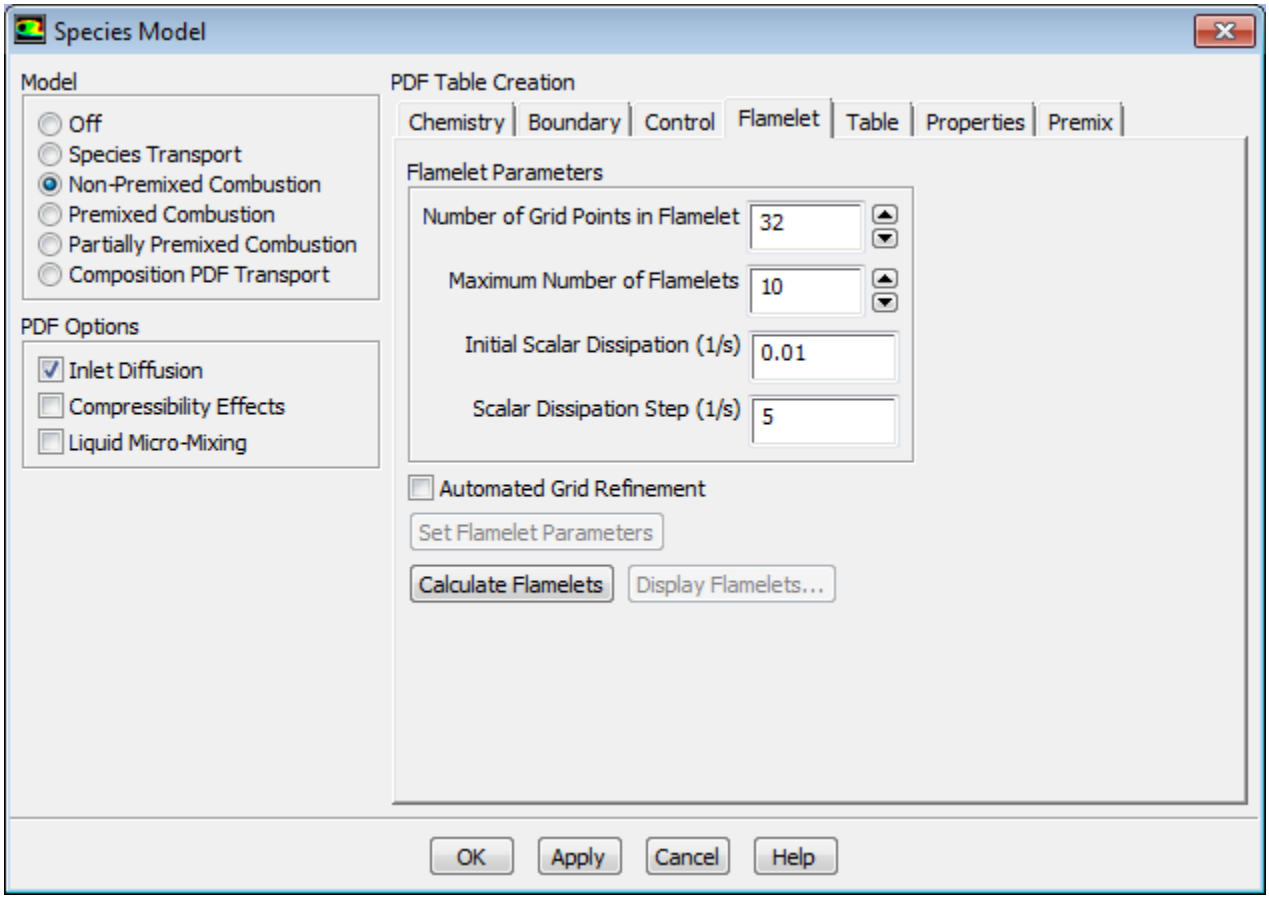
Figure 15.3: Defining Chemical Boundary Species



5. (steady diffusion flamelet model only) If you are generating flamelets, compute the flamelet state relationships of species mass fractions, density, and temperature as a function of mixture fraction and scalar dissipation (Figure 15.4: Calculating Steady Diffusion Flamelets (p. 1030)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

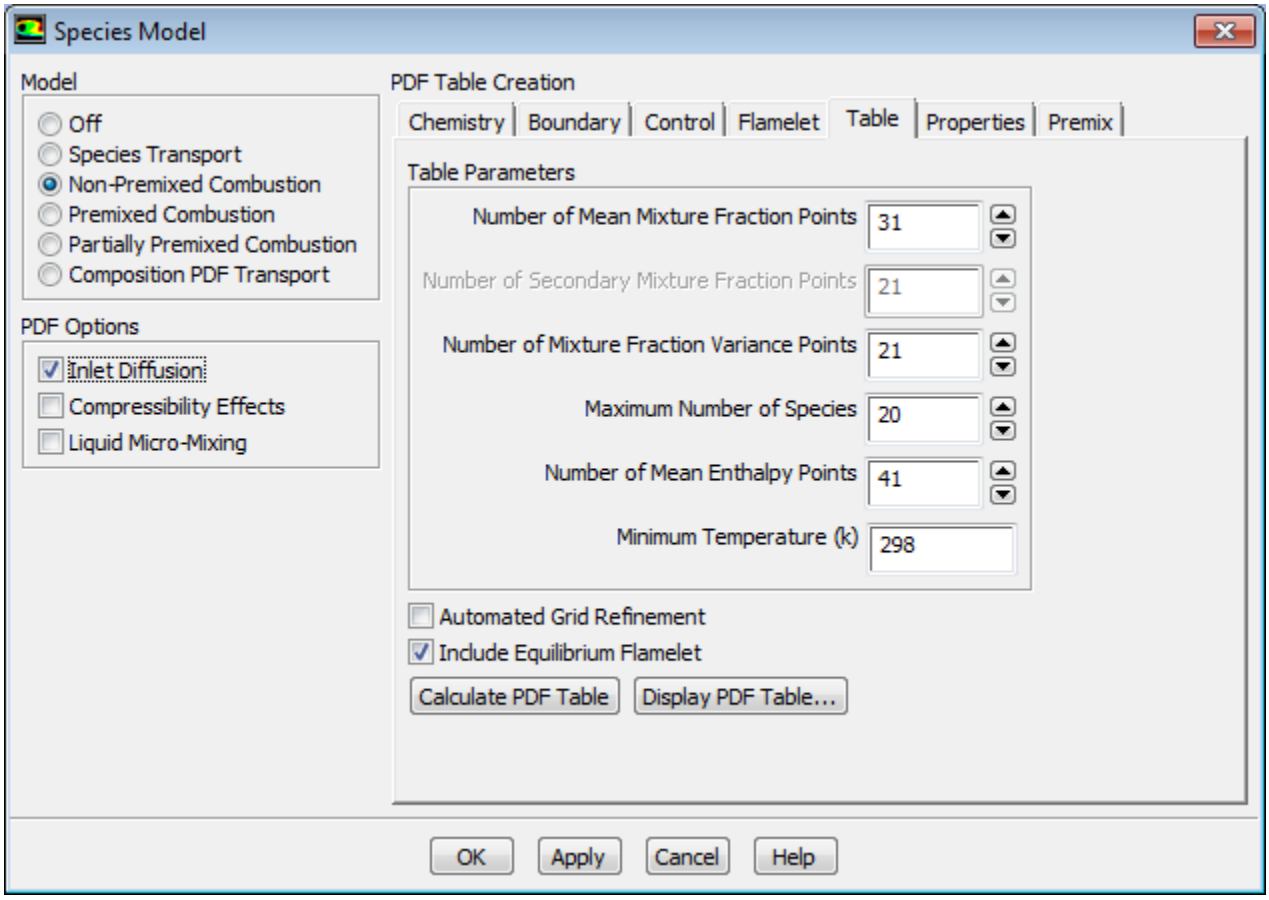
Figure 15.4: Calculating Steady Diffusion Flamelets



6. Compute the final chemistry look-up table, containing mean values of species fractions, density, and temperature as a function of mean mixture fraction, mixture fraction variance, and possibly enthalpy and scalar dissipation. The contents of this look-up table will reflect your preceding inputs describing the turbulent reacting system (Figure 15.5: Calculating the Chemistry Look-Up Table (p. 1031)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.5: Calculating the Chemistry Look-Up Table



The look-up table is the stored result of the integration of Equation 8.16 (or Equation 8.24) and Equation 8.18 (in the Theory Guide). The look-up table will be used in ANSYS Fluent to determine mean species mass fractions, density, and temperature from the values of mean mixture fraction ($\overline{f}$), mixture fraction variance ($\overline{f'^2}$), and possibly mean enthalpy ($\overline{H}$) and mean scalar dissipation ($\overline{\chi}$) as they are computed during the ANSYS Fluent calculation of the reacting flow. See Look-Up Tables for Adiabatic Systems and Figure 8.8: Visual Representation of a Look-Up Table for the Scalar (Mean Value of Mass Fractions, Density, or Temperature) as a Function of Mean Mixture Fraction and Mixture Fraction Variance in Adiabatic Single-Mixture-Fraction Systems and Figure 8.10: Visual Representation of a Look-Up Table for the Scalar as a Function of Mean Mixture Fraction and Mixture Fraction Variance and Normalized Heat Loss/Gain in Non-Adiabatic Single-Mixture-Fraction Systems in the Theory Guide.

For a problem that includes a secondary stream (and, therefore, a second mixture fraction), you will perform the first two steps listed above for the single-mixture-fraction approach and then prepare a look-up table of instantaneous properties using Equation 8.12 or Equation 8.14 in the Theory Guide.

15.2. Setting Up the Equilibrium Chemistry Model

In the equilibrium chemistry model, the concentrations of species of interest are determined from the mixture fraction using the assumption of chemical equilibrium (see Non-Premixed Combustion and Mixture Fraction Theory in the Theory Guide). With this model, you can include the effects of intermediate species and dissociation reactions, producing more realistic predictions of flame temperatures than the Eddy-Dissipation model. When you choose the equilibrium chemistry option, you will have the opportunity to use the rich flammability limit (RFL) option.

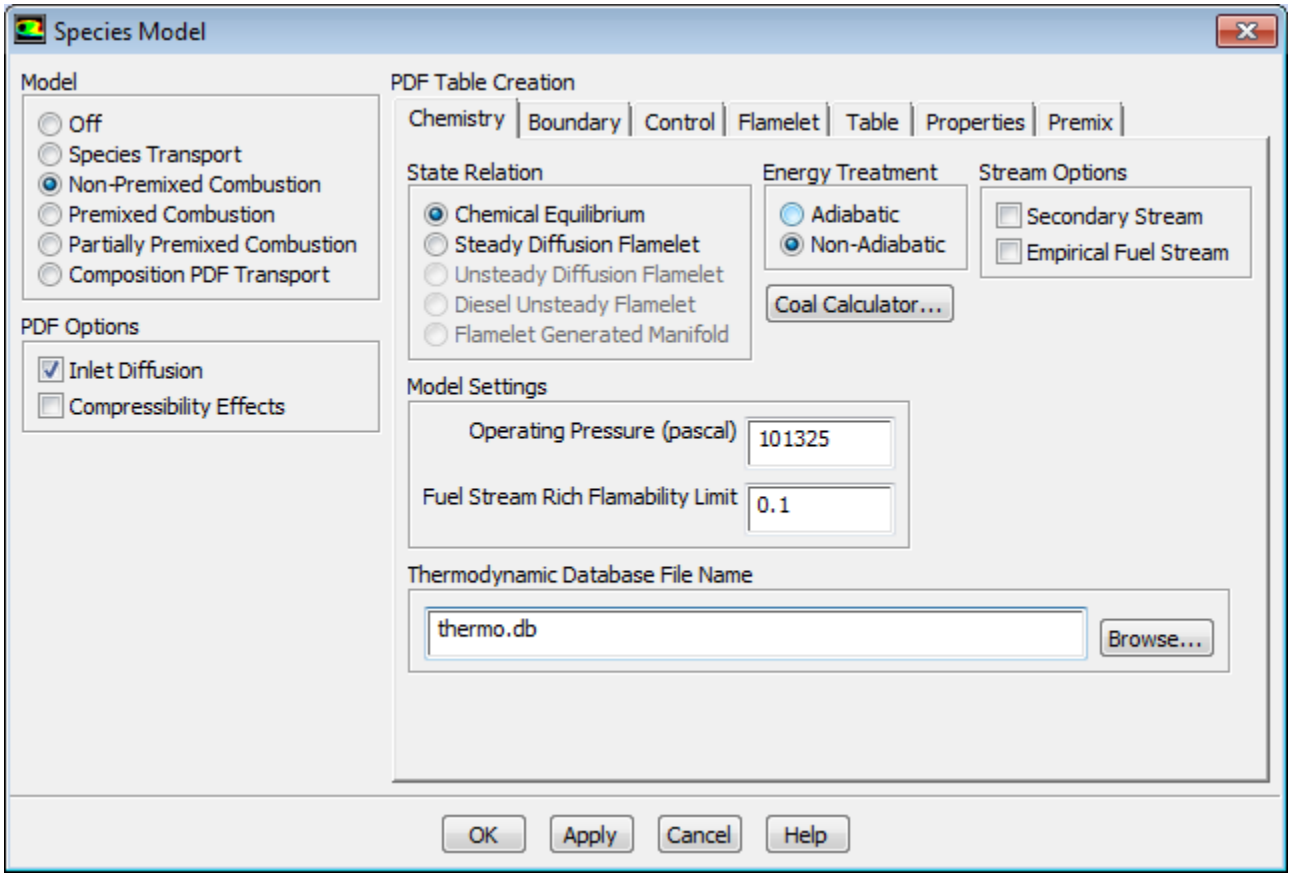
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To enable the equilibrium chemistry model

1. Select **Non-Premixed Combustion** in the **Species Model** dialog box.

2. Select **Chemical Equilibrium** in the **Chemistry** tab of the **Species Model** dialog box.

Figure 15.6: The Species Model Dialog Box (Chemistry Tab)



For additional information, see the following sections:

15.2.1. Choosing Adiabatic or Non-Adiabatic Options
15.2.2. Specifying the Operating Pressure for the System
15.2.3. Enabling a Secondary Inlet Stream
15.2.4. Choosing to Define the Fuel Stream(s) Empirically
15.2.5. Enabling the Rich Flammability Limit (RFL) Option

15.2.1. Choosing Adiabatic or Non-Adiabatic Options

You should use the non-adiabatic modeling option if your problem definition in ANSYS Fluent will include one or more of the following:

- radiation or wall heat transfer
- multiple fuel inlets at different temperatures
- multiple oxidant inlets at different temperatures
- liquid fuel, coal particles, and/or heat transfer to inert particles

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that the adiabatic model is a simpler model involving a two-dimensional look-up table in which scalars depend only on $\overline{f}$ and $\overline{f'^2}$ (or on f_{fuel} and p_{sec}). If your model is defined as adiabatic, you will not need to solve the energy equation in ANSYS Fluent and the system temperature will be determined directly from the mixture fraction and the fuel and oxidant inlet temperatures. The non-adiabatic case will be more complex and more time-consuming to compute, requiring the generation of three-dimensional look-up tables. However, the non-adiabatic model option allows you to include the types of reacting systems described above.

Select **Adiabatic** or **Non-Adiabatic** in the **Chemistry** tab of the **Species Model** dialog box.

15.2.2. Specifying the Operating Pressure for the System

The system **Operating Pressure** is used to calculate density using the ideal gas law. For non-adiabatic simulations, the **Compressibility Effects** under **PDF Options** can be enabled to account for cases where substantial pressure changes occur in time and/or space. In such cases it is assumed that the species mass fractions do not change with pressure, and the density is calculated as

$$\overline{\rho}=\rho_{op}\frac{\overline{p}}{\overline{p}_{op}} \tag{15.1}$$

where ρ_{op} is the density at the specified **Operating Pressure** (p_{op}), and $\overline{p}$ is the local mean pressure in an ANSYS Fluent cell.

When the **Compressibility Effects** option is enabled, the flow operating pressure (set in the **Operating Conditions** dialog box) can differ from the Non-Premixed model operating pressure. To distinguish this difference, the **Operating Pressure** name tag in the **Species Model** dialog box changes to **Equilibrium Operating Pressure** when the compressibility effects option is enabled.

See Solution Strategies for Non-Premixed Modeling (p. 1081) for details about enabling compressibility effects.

15.2.3. Enabling a Secondary Inlet Stream

If you are modeling a system consisting of a single fuel and a single oxidizer stream, you do not need to enable a secondary stream in your PDF calculation. As discussed in Definition of the Mixture Fraction in the Theory Guide, a secondary stream should be enabled if your PDF reaction model will include any of the following:

- two dissimilar gaseous fuel streams
  - In these simulations, the fuel stream defines one of the fuels and the secondary stream defines the second fuel.
- mixed fuel systems of dissimilar gaseous and liquid fuel
  - In these simulations, the fuel stream defines the gaseous fuel and the secondary stream defines the liquid fuel (or vice versa).
- mixed fuel systems of dissimilar gaseous and coal fuels
  - In these simulations, you can use the fuel stream or the secondary stream to define either the coal or the gaseous fuel. See Modeling Coal Combustion Using the Non-Premixed Model (p. 1046) regarding coal combustion simulations with the non-premixed combustion model.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- mixed fuel systems of coal and liquid fuel
  - In these simulations, you can use the fuel stream or the secondary stream to define either the coal or the liquid fuel. See Modeling Coal Combustion Using the Non-Premixed Model (p. 1046) regarding coal combustion simulations with the non-premixed combustion model.
- coal combustion
  - Coal combustion can be more accurately modeled by using a secondary stream to track the distinct volatile and char off-gases. The fuel stream must define the char and the secondary stream must define the volatile components of the coal. See Modeling Coal Combustion Using the Non-Premixed Model (p. 1046) regarding coal combustion simulations with the non-premixed combustion model.
- a single fuel with two dissimilar oxidizer streams
  - In these simulations, the fuel stream defines the fuel, the oxidizer stream defines one of the oxidizers, and the secondary stream defines the second oxidizer.

To include a secondary stream in your model, turn on the **Secondary Stream** option under **Stream Options** in the **Chemistry** tab.

Important

Using a secondary stream can substantially increase the calculation time for your simulation since the multi-dimensional PDF integrations are performed at run time. Alternatively, ANSYS Fluent can perform a full tabulation of the PDF integrations, as detailed in Full Tabulation of the Two-Mixture-Fraction Model (p. 1067).

15.2.4. Choosing to Define the Fuel Stream(s) Empirically

The empirical fuel option provides an alternative method for defining the composition of the fuel or secondary stream when the individual species components of the fuel are unknown. That is, you will define the elemental fraction *not* the individual species. When this option is disabled, you will define the chemical species that are present in each stream and the mass or mole fraction of each species, as described in Defining the Stream Compositions (p. 1042). The option for defining an empirical fuel stream is particularly useful for coal combustion simulations (see Modeling Coal Combustion Using the Non-Premixed Model (p. 1046)) or for simulations involving other complex hydrocarbon mixtures.

To define a fuel or secondary stream empirically

1. Turn on the **Empirical Fuel Stream** option under **Stream Options** in the **Chemistry** tab of the **Species Model** dialog box. If you have a secondary stream, enable the **Empirical Secondary Stream** option, or both as appropriate.
2. Specify the appropriate lower heating value (for example **Empirical Fuel Lower Caloric Value**, **Empirical Secondary Lower Caloric Value**), specific heat (**Empirical Fuel Specific Heat**, **Empirical Secondary**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specific Heat), and molecular weight (**Empirical Fuel Molecular Weight**, **Empirical Secondary Molecular Weight**) for each empirically defined stream.

Important

The empirical definition option is available only with the full equilibrium chemistry model. It cannot be used with the rich flammability limit (RFL) option or the steady and unsteady diffusion flamelet models, since equilibrium calculations are required for the determination of the fuel composition.

Important

The empirical fuel and secondary molecular weights are only required if your empirical streams are entering the domain via an inlet boundary, or if you are using the partially premixed model. If you are using the non-premixed model and the empirically defined streams originate from the dispersed phase (for example, if you are modeling coal or liquid fuel combustion) the molecular weights are not required for the computation.

15.2.5. Enabling the Rich Flammability Limit (RFL) Option

You can define a rich limit on the mixture fraction when the equilibrium chemistry option is used. Input of the rich limit is accomplished by specifying a value of the **Rich Flammability Limit** for the appropriate **Fuel Stream**, **Secondary Stream**, or both. You will not be allowed to specify the **Rich Flammability Limit** if you have used the empirical definition option for fuel composition.

ANSYS Fluent will compute the composition at the rich limit using equilibrium. For mixture fraction values above this limit, ANSYS Fluent will suspend the equilibrium chemistry calculation and will compute the composition based on mixing, but not burning, of the fuel with the composition at the rich limit. A value of 1.0 for the rich limit implies that equilibrium calculations will be performed over the full range of mixture fraction. When you use a rich limit that is less than 1.0, equilibrium calculations are suspended whenever f, f_{fuel}, or f_{sec} exceeds the limit. This RFL model is often more accurate than the assumption of chemical equilibrium for rich mixtures, and also avoids complex equilibrium calculations, speeding up the preparation of the look-up tables. An RFL value of approximately twice the stoichiometric mixture fraction is appropriate.

For the **Secondary Stream**, the rich flammability limit controls the equilibrium calculation for the secondary mixture fraction. If your secondary stream is not a fuel, you should use an RFL value of 1. A value of 1.0 for the rich limit implies that equilibrium calculations will be performed over the full range of mixture fraction. When you input a rich limit that is less than 1.0, equilibrium calculations are suspended whenever f_{sec} exceeds the limit. (Note that it is the secondary mixture fraction f_{sec} and not the partial fraction p_{sec} that is used here.)

Important

Experimental studies and reviews [12] (p. 2887), [106] (p. 2892) have shown that although the fuel-lean flame region approximates thermodynamic equilibrium, non-equilibrium kinetics will prevail under fuel-rich conditions. Therefore, for non-empirically defined fuels, the RFL model is strongly recommended.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

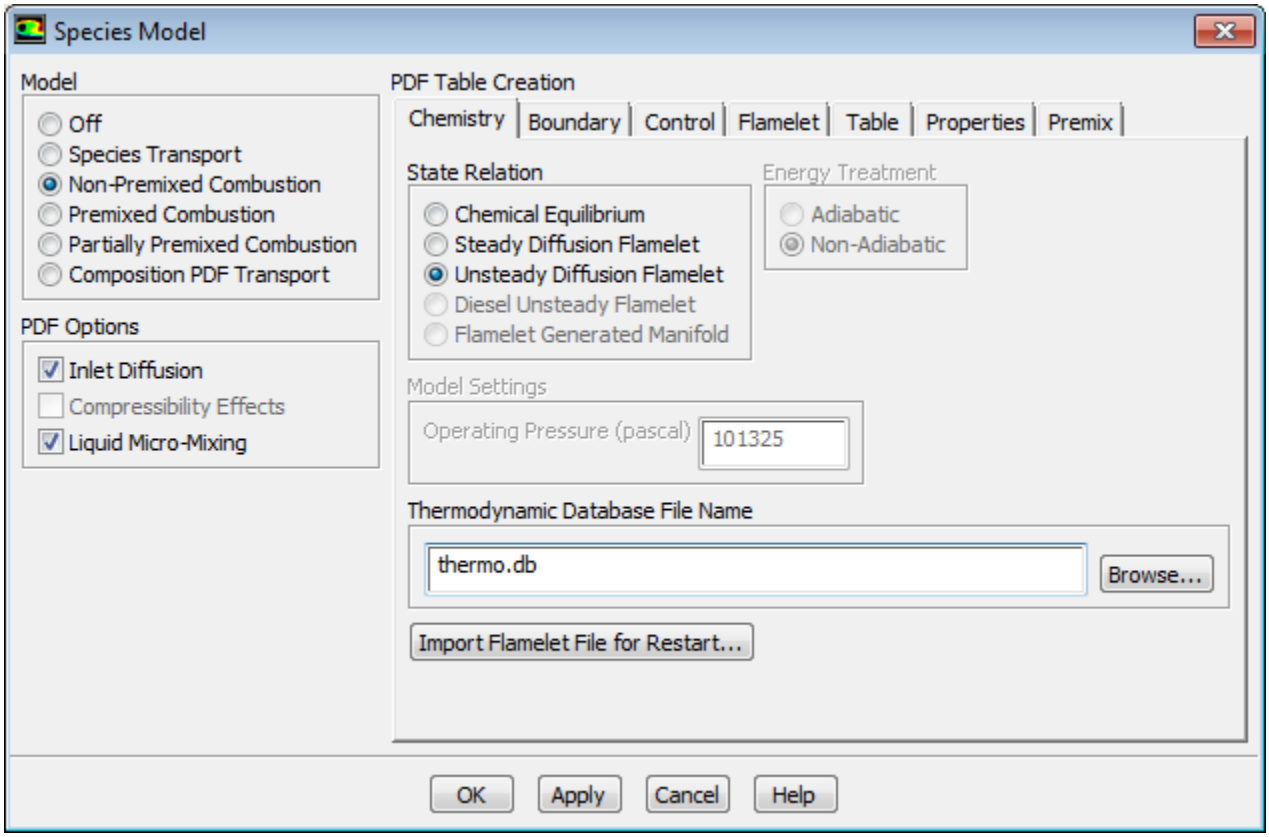
15.3. Setting Up the Steady and Unsteady Diffusion Flamelet Models

To enable the diffusion flamelet models

1. Select **Non-Premixed Combustion** in the **Species Model** dialog box.

2. Select **Steady Diffusion Flamelet** or **Unsteady Diffusion Flamelet** in the **Chemistry** tab of the **Species Model** dialog box. See Using the Unsteady Diffusion Flamelet Model (p. 1037).

Figure 15.7: The Chemistry Tab for the Unsteady Diffusion Flamelet Model



For additional information, see the following sections:

15.3.1. Choosing Adiabatic or Non-Adiabatic Options
15.3.2. Specifying the Operating Pressure for the System
15.3.3. Specifying a Chemical Mechanism File for Flamelet Generation
15.3.4. Importing a Flamelet
15.3.5. Using the Unsteady Diffusion Flamelet Model
15.3.6. Using the Diesel Unsteady Laminar Flamelet Model
15.3.7. Resetting Diesel Unsteady Flamelets

15.3.1. Choosing Adiabatic or Non-Adiabatic Options

Select **Adiabatic** or **Non-Adiabatic** in the **Chemistry** tab of the **Species Model** dialog box. See the discussion in Choosing Adiabatic or Non-Adiabatic Options (p. 1032) about the two options.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.3.2. Specifying the Operating Pressure for the System

The system **Operating Pressure** is used to calculate density using the ideal gas law. When the **Compressibility Effects** option is enabled, the name **Operating Pressure** is changed to **Equilibrium Operating Pressure** since the non-premixed combustion model operating pressure can differ from the flow operating pressure. Specifying the Operating Pressure for the System (p. 1033) provides more information about this value.

You can use the steady or unsteady diffusion flamelet model for reactions in liquid systems. To do so, enable **Liquid Micro-Mixing** under **PDF Options**. The **Liquid Micro-Mixing** option is discussed in detail in Liquid Reactions in the Theory Guide.

15.3.3. Specifying a Chemical Mechanism File for Flamelet Generation

If you are generating a flamelet file yourself, you will need to read in the chemical kinetic mechanism and thermodynamic data. The mechanism and thermodynamic data must be in CHEMKIN format [52] (p. 2889).

To read in a CHEMKIN mechanism, select the **Create Flamelet** option in the **Chemistry** tab of the **Species Model** dialog box and click the **Import CHEMKIN Mechanism...** button. Using the **CHEMKIN Mechanism Import** dialog box, import the gas-phase CHEMKIN files into ANSYS Fluent as described in Importing a Volumetric Kinetic Mechanism in CHEMKIN Format (p. 967). Note that for flamelet generation, only a gas-phase kinetic mechanism and a thermodynamic database are required.

Note that the import is limited to mechanisms with 700 or fewer species.

15.3.4. Importing a Flamelet

To import an existing flamelet file

1. Select the **Import Flamelet** option in the **Chemistry** tab of the **Species Model** dialog box.
2. (steady diffusion flamelet only) Select either **Standard** or **CFX-RIF** format under **File Type**.
3. Click the **Import Flamelet File...** button. In The Select File Dialog Box (p. 19), select the file (for a single flamelet) or files (for multiple flamelets) to be read in to ANSYS Fluent.

After you have completed this step, you can skip ahead to the **Table** tab of the **Species Model** dialog box (see Calculating the Look-Up Tables (p. 1063)).

15.3.5. Using the Unsteady Diffusion Flamelet Model

The unsteady diffusion flamelet model can only be enabled from a valid steady-state, steady diffusion flamelet solution. When enabled, the unsteady diffusion flamelet model will change this case to unsteady and postprocess marker probability equations on the frozen flow field. You should hence ensure that the starting steady-state, steady diffusion flamelet solution is fully converged.

When the **Unsteady Diffusion Flamelet** is enabled in the **Chemistry** tab, the **Import Flamelet File for Restart...** button appears in the dialog box, allowing you to run the simulation from a previously saved case, data and unsteady flamelet file.

The Unsteady Diffusion Flamelet Model requires four user inputs in the **Flamelet** tab:

- The **Number of Grid Points in Flamelet**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The **Mixture Fraction Lower Limit for Initial Probability**. The initial condition of the marker probability field is unity for all mean mixture fractions above the **Mixture Fraction Lower Limit for Initial Probability**, and zero for mean mixture fractions below it. Note that this should be specified to be greater than the stoichiometric mixture fraction.

- **Maximum Scalar Dissipation**. Flamelets may extinguish at high scalar dissipations because diffusion in the flamelet overwhelms reaction. It is possible to have unrealistically high modeled scalar dissipation in the 2D or 3D ANSYS Fluent simulations, which gets transferred to the 1D unsteady flamelet. In order to avoid excessive diffusion in the 1D unsteady flamelet, the instantaneous scalar dissipation in the 1D flamelet is limited to the specified **Maximum Scalar Dissipation**.

- **Courant Number**. The time step for the unsteady probability marker equation is calculated automatically by ANSYS Fluent based on the **Courant Number**. Larger values imply fewer time steps to convect/diffuse the marker probability out of the domain, but also results in a larger numerical error. The **Courant Number** should be small enough so that the unsteady flamelet mean mass fractions are unchanged with any smaller **Courant Number**. The default value of 1 should be sufficient for most applications.

- **Number of Flamelets**. The number of unsteady laminar flamelets that ANSYS Fluent will generate during the run. The marker probability equation Equation 8.56 in the *Fluent Theory Guide* will be solved for each flamelet.

When these inputs have been set, clicking the **Initialize Unsteady Flamelet Probability** button initializes the marker probability equation for each flamelet, automatically enabling the **Unsteady** solver, while disabling all equations except the **Unsteady Flamelet Probability** equation in the **Solution Controls** task page. This initialization in the **Flamelet** tab also sets the **Time Step Size** in the **Run Calculation** task page.

Important

- Do not initialize your solution from the tree or the **Solution Initialization** task page. Note that you are postprocessing a probability field on the frozen steady-state flow field, and by clicking the **Initialize Unsteady Flamelet Probability** button, you have already initialized the probability marker field.

- If you disable the **Unsteady Diffusion Flamelet** model and you want to revert to solving a steady diffusion flamelet simulation, make sure you enable **Steady** (either in the **General** task page or in the tree from the **Setup**/**General**/ **Analysis Type** tree item) and enable all the equations (in the **Solution Controls** task page or in the tree from the **Solution**/**Solution Controls** tree item).

15.3.6. Using the Diesel Unsteady Laminar Flamelet Model

The diesel unsteady laminar flamelet model can only be enabled when conditions for compression-ignition are met:

- The **Transient** solver is selected in the **General** task page (or in the tree from the **Setup**/**General**/ **Analysis Type**).

- The **In-Cylinder** dynamic mesh is enabled.

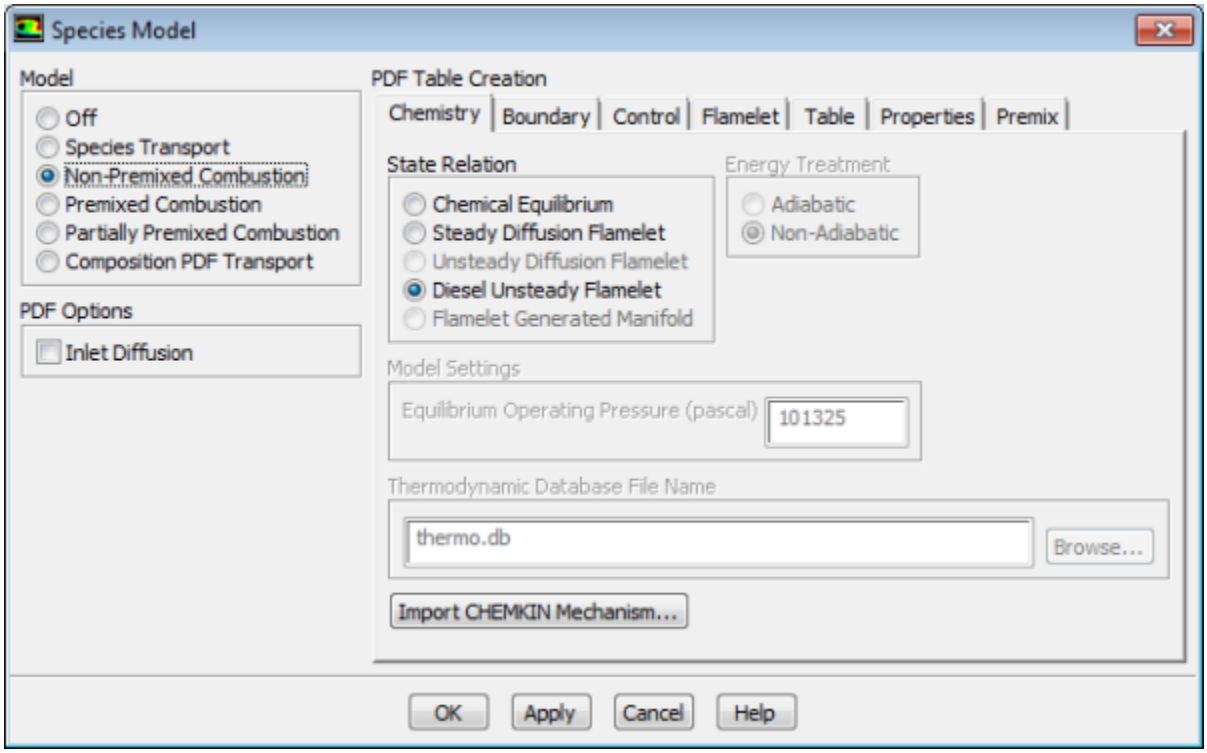
- The Discrete Phase model option **Interaction with Continuous Phase** is selected.

The basic steps for setting the diesel unsteady laminar flamelet models are as follows.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the **Chemistry** tab, select **Diesel Unsteady Flamelet**.

Figure 15.8: The Enabled Diesel Unsteady Flamelet Model



2. If a detailed chemical mechanism containing kinetic reactions appropriate for compression ignition has not yet been defined in your case, you can import a mechanism in CHEMKIN format as described in Importing a Volumetric Kinetic Mechanism in CHEMKIN Format (p. 967).

   The mechanism can include pollutant formation reactions as well if you are interested in modeling emissions.

3. In the **Boundary** tab, define the stream compositions as described in Defining the Stream Compositions (p. 1042).

4. In the **Flamelet** tab, set the following flamelet parameters.

   a. Set the **Number of Grid Points in Flamelet**.

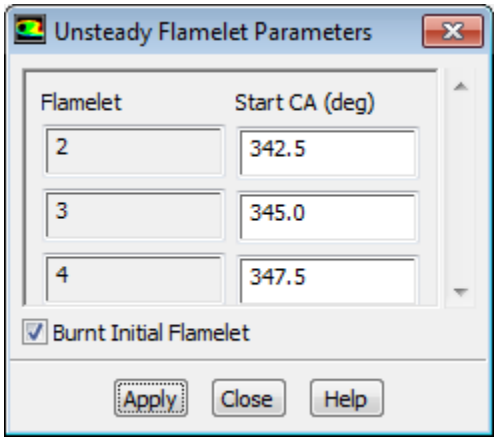
   b. Set the **Number of Unsteady Flamelets** that ANSYS Fluent will generate during simulation.

   c. (for multiple unsteady flamelets) Set the flamelet start times.

      ANSYS Fluent automatically sets the start time for the first flamelet, but you must set the start time for each consecutive flamelet using the **Unsteady Flamelet Parameters** dialog box. Open it by clicking **Set Flamelet Parameters** and enter the start time for each flamelet either in seconds or in crank angles if the dynamic mesh is enabled.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.9: The Unsteady Flamelet Parameters Dialog Box



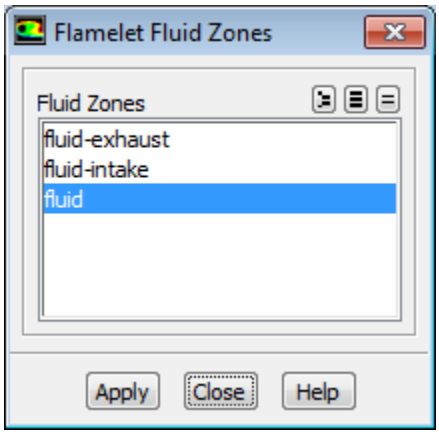
ANSYS Fluent starts simulation with only one flamelet, and then it automatically introduces new flamelets into the reacting domain at the times you have specified.

d. The default initial condition for an unsteady flamelet is unburnt. ANSYS Fluent provides the **Burnt Initial Flamelet** option that allows you to set the initial flamelet condition to a chemical equilibrium burnt state. This option is useful if you are modeling internal combustion engines where residual gases may be present in the cylinder before the spray is injected, which would be incorrectly modeled by the unburnt state. Note that the **Burnt Initial Flamelet** is only used at case initialization.

e. (optional) Specify the flamelet fluid zones.

ANSYS Fluent calculates diesel unsteady flamelets using the zone-averaged pressure and scalar dissipation at every time step. By default, the averaging is performed over all fluid zones in the domain, but you can also select and/or deselect the fluid zones using the **Flamelet Fluid Zones** dialog box. Open this dialog box by clicking **Set Flamelet Fluid Zones** and select the fluid zones to be used for calculating average pressure and scalar dissipation. If no fluid zone is selected, ANSYS Fluent will compute domain average pressure and scalar dissipation using all fluid zones.

Figure 15.10: The Flamelet Fluid Zones Dialog Box



Note

For internal combustion cases, it is recommended that you select the cylinder fluid zones and deselect the intake and exhaust fluid zones.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that ANSYS Fluent calculates flamelets at every time step of the run. For this reason, the option to calculate the flamelets as a preprocessing step before running your simulation is unavailable, and **Calculate Flamelets** appears dimmed.

5. In the **Table** tab, set the PDF table parameters as described in Calculating the Look-Up Tables (p. 1063).

   Note that ANSYS Fluent calculates PDF table at every time step of the run. For this reason, the option to calculate the PDF table as a preprocessing step before running your simulation is unavailable, and **Calculate PDF Table** appears dimmed.

15.3.6.1. Recommended Settings for Internal Combustion Engines

When setting up and using the Diesel Unsteady Flamelet model for internal combustion engine simulations, the following recommendations apply:

1. Number of Flamelets

   a. You must specify at least two diesel unsteady flamelets. ANSYS Fluent will use the first flamelet to model trapped burnt gases from the previous cycle. The second flamelet will start at the crank angle (CA) of fuel injection, specified in the **Unsteady Flamelet Parameters** dialog box (see Figure 15.10: The Flamelet Fluid Zones Dialog Box (p. 1040)).

   b. To model split injections where an initial fuel mass is injected and burns before the main fuel injection, three or more unsteady flamelets are required.

2. Flamelet Initialization

   By default, the flamelet is initialized as mixed-but-unburnt. However, in all practical scenarios there is always some trapped gas remaining inside the cylinder. Therefore, it is recommended that you use the **Burnt Initial Flamelet** option in the **Unsteady Flamelet Parameters** dialog box (see Figure 15.10: The Flamelet Fluid Zones Dialog Box (p. 1040)). When this option is selected, ANSYS Fluent performs a constant temperature equilibrium calculation and sets the initial flamelet condition to a chemical equilibrium burnt state.

3. Multi-cycle simulations

   a. To accurately model multiple cycles of internal combustion engines, the flamelets must be reset at the end of each cycle. This is performed by defining the **Diesel Unsteady Flamelet Reset** event, typically at the specified crank angle, just before the inlet valve opens. Refer to the Resetting Diesel Unsteady Flamelets (p. 1041) for details. This approach is recommended for modeling the EGR trapped gases with the first burnt unsteady flamelet.

   b. If you are using the Inert (EGR) model in order to track the trapped inert mixture, you need to define the **Inert EGR Reset** event at the specified crank angle just before the inlet valve opens. See Resetting Inert EGR (p. 1077) for details.

15.3.7. Resetting Diesel Unsteady Flamelets

In order to simulate multiple cycles in internal combustion engines, flamelets should be reset at the end of every cycle. In addition, the burned trapped gases must be modeled, which can be done in one of the two ways. The first and recommended approach is to use the **Diesel Unsteady Flamelet Reset** option. The second approach is to use the inert (EGR) model and the **Inert EGR Reset** option. You can access the **Diesel Unsteady Flamelet Reset** and **Inert EGR Reset** options via the **Dynamic Mesh Events** dialog box. There, you need to set the crank angle at which this event occurs (usually shortly before

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the inlet valves open) and the participating fluid zones (usually only the combustion chamber and not the intake and exhaust port zones).

When the **Diesel Unsteady Flamelet Reset** event is executed, all flamelets are deleted and a new flamelet is introduced with a state set to the probability-weighted average condition of all flamelets present before reset. The other new flamelets are introduced during a new cycle in a similar fashion to that described in Using the Diesel Unsteady Laminar Flamelet Model (p. 1038).

When the inert EGR reset event is executed with the diesel unsteady flamelet model, the burnt gas in the selected Inert EGR Reset zones is converted to inert, all flamelets are deleted, and a new unburnt flamelet is introduced into the domain.

Note

The **Diesel Unsteady Flamelet Reset** option is available only when the selected number of flamelets is greater than one.

15.4. Defining the Stream Compositions

In modeling a non-premixed combustion problem, you will only specify the boundary species (that is, the fuel, oxidizer, and if necessary, secondary stream species). The intermediate and product species will be determined automatically.

ANSYS Fluent provides you with an initial list of common boundary species (ch4, h2, jet-a<g>, n2 and o2). If your fuel and/or oxidizer is composed of different species, you can add them to the boundary **Species** list. All boundary species must exist in the chemical database and you must enter their names in the same format used in the database, otherwise an error message will be issued.

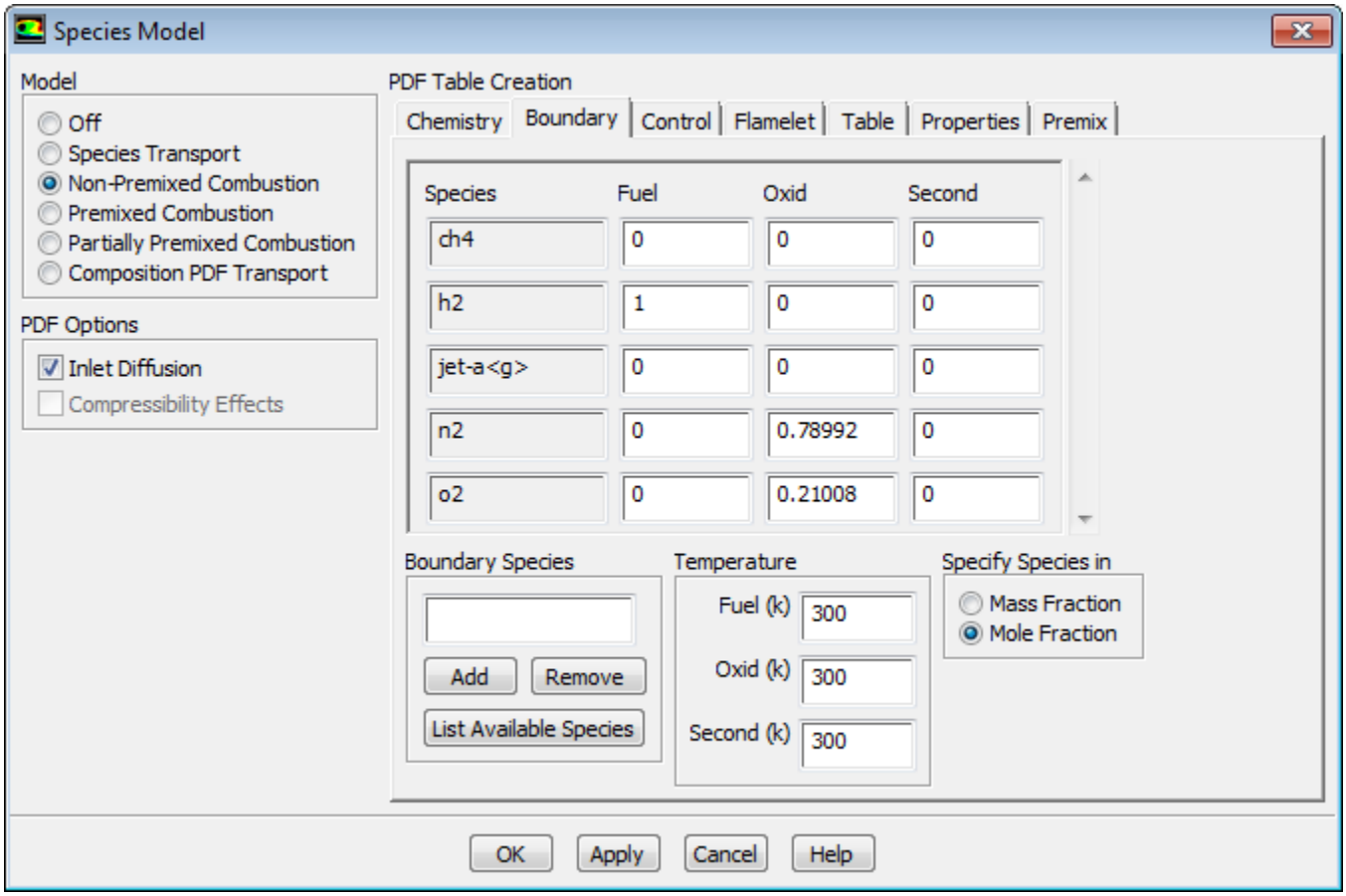
After defining the boundary species that will be considered in the reaction system, you must define their mole or mass fractions at the fuel and oxidizer inlets and at the secondary inlet, if one exists. (If you choose to define the fuel or secondary stream composition empirically, you will instead enter the parameters described at the end of this section.) For the example shown in Figure 8.12: Chemical Systems That Can Be Modeled Using a Single Mixture Fraction in the Theory Guide, for example, the fuel inlet consists of 60% CH_4, 20% CO, 10% CO_2, and 10% C_3H_8.

Finally, the inlet stream temperatures of your reacting system are required for construction of the look-up table and computation of the equilibrium chemistry model.

For the equilibrium chemistry model, the species names are entered using the **Boundary** tab in the **Species Model** dialog box (Figure 15.11: The Species Model Dialog Box (Boundary Tab) (p. 1043)). If you are generating a steady or unsteady diffusion flamelet, the list of boundary species will be automatically filled as all the species in the CHEMKIN mechanism, and you will be unable to change these.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.11: The Species Model Dialog Box (Boundary Tab)



The steps for adding new species and defining their compositions are as follows:

1. (chemistry equilibrium chemistry model only) If your fuel, oxidizer, or secondary streams are composed of species other than the default species list, type the chemical formula (for example, `so2` or `S02` for SO_2) under **Boundary Species** and click **Add**. The species will be added to the **Species** list. Continue in this manner until all of the boundary species you want to include are shown in the **Species** list.

   To remove a species from the list, type the chemical formula under **Boundary Species** and click **Remove**. To print a list of all species in the thermodynamic database file (`thermo.db`) in the console window, click **List Available Species**.

2. (non-equilibrium chemistry models) Optionally, select the boundary species as described in Overview of the Problem Setup Procedure (p. 1026).

3. Under **Specify Species In**, specify whether you want to enter the **Mass Fraction** or **Mole Fraction**. **Mass Fraction** is the default.

4. For each relevant species in the **Species** list, specify its mass or mole fraction for each stream (**Fuel**, **Oxid**, or **Second** as appropriate) by entering values in the table. Note that if you change from **Mass Fraction** to **Mole Fraction** (or vice versa), all values will be automatically converted if they sum to 0 or 1, so be sure that you are entering either all mass fractions or all mole fractions as appropriate. If the values do not sum to 0 or 1, an error will be issued.

5. Under **Temperature**, specify the following inputs:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fuel

is the temperature of the fuel inlet in your model. In adiabatic simulations, this input (together with the oxidizer inlet temperature) determines the inlet stream temperatures that will be used by ANSYS Fluent. In non-adiabatic systems, this input should match the inlet thermal boundary condition that you will use in ANSYS Fluent (although you will enter this boundary condition again in the ANSYS Fluent session). If your ANSYS Fluent model will use liquid fuel or coal combustion, define the inlet fuel temperature as the temperature at which vaporization/devolatilization begins (that is, the **Vaporization Temperature** specified for the discrete-phase material—see Setting Material Properties for the Discrete Phase (p. 1313)). For such non-adiabatic systems, the inlet temperature will be used only to adjust the look-up table grid (for example, the discrete enthalpy values for which the look-up table is computed). Note that if you have more than one fuel inlet, and these inlets are not at the same temperature, you must define your system as non-adiabatic. In this case, you should enter the fuel inlet temperature as the value at the dominant fuel inlet.

Oxid

is the temperature of the oxidizer inlet in your model. The issues raised in the discussion of the input of the fuel inlet temperature (directly above) pertain to this input as well.

Second

is the temperature of the secondary stream inlet in your model. (This item will appear only when you have defined a secondary inlet.) The issues raised in the discussion of the input of the fuel inlet temperature (directly above) pertain to this input as well.

For additional information, see the following sections:

15.4.1. Setting Boundary Stream Species
15.4.2. Modifying the Database
15.4.3. Composition Inputs for Empirically-Defined Fuel Streams
15.4.4. Modeling Liquid Fuel Combustion Using the Non-Premixed Model
15.4.5. Modeling Coal Combustion Using the Non-Premixed Model

15.4.1. Setting Boundary Stream Species

In combustion, a large number of intermediate and product species may be produced from a small number of initial boundary species. In ANSYS Fluent you must specify only the species composition of your boundary species in the fuel, oxidizer, and (if appropriate) secondary streams. ANSYS Fluent will calculate all intermediate and product species automatically. The following suggestions may be helpful in the definition of the system chemistry:

- For coal combustion, char in the coal should be represented by C(s).

  Important

  Care should be taken to distinguish atomic carbon, C, from solid carbon, C(s). Atomic carbon should be selected only if you are using the empirically-defined input method.

- If your fuel composition is known empirically (for example, $C_{0.9}H_3O_{0.2}$), use the option for an empirically-defined stream (see below).

- If you want to include the sulfur that may be present in a hydrocarbon fuel, note that this may hinder the convergence of the equilibrium solver, especially if the concentration of sulfur is small. It is therefore recommended that you include sulfur in the calculation only if it is present in considerable quantities.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.4.1.1. Including Condensed Species

In addition to gaseous species, liquid and solid species can be included in the chemistry calculations. They are often indicated by an "l" or an "s" in parentheses after the species name. If you add a condensed species to the equilibrium chemical system, its density will be retrieved from ANSYS Fluent's chemical property database file `propdb.scm` if you are using the thermodynamic database file `thermo.db` that is also supplied with ANSYS Fluent. If you are using a custom thermodynamic database file and want to include a condensed species in the equilibrium system that does not exist in `propdb.scm`, a density of 1000 kg/m^3 will be assumed. The condensed species density can be changed in the Create/Edit Materials Dialog Box (p. 2302) after the PDF table has been calculated. If you modify the condensed species density in this manner, you will then need to recalculate the PDF table.

15.4.2. Modifying the Database

If you want to include a new species in your reacting system that is not available in the chemical database, you can add it to the database file, `thermo.db`. The format for `thermo.db` is detailed in [52] (p. 2889). If you choose to modify the standard database file, you should create copies of the original file.

15.4.3. Composition Inputs for Empirically-Defined Fuel Streams

As mentioned in Defining the Problem Type (p. 1026), you can define the composition of a fuel stream (that is, the standard fuel or a secondary fuel) empirically. For an empirically-defined stream, you will need to enter the atomic mass or mole fractions in addition to the inputs for lower caloric (heating) value of the fuel and the mean specific heat of the fuel that were described previously.

The heat of formation of an empirically defined stream is calculated from the heating value and the atomic composition. The fuel inlet temperature and fuel specific heat are used to calculate the sensible enthalpy. The molecular weight is used for the computation of the unburnt stream density. Note that the unburnt density is only required if the stream enters via an inlet boundary, or if you are using the partially-premixed model.

When an empirically-defined fuel or secondary stream is specified in the **Chemistry** tab (equilibrium chemistry model only) of the **Species Model** dialog box, you must specify the atom fractions of C, H, O, N, and S in that stream instead of the species mass or mole fractions. To avoid confusion, the species fraction inputs for an empirically-defined stream will be grayed out in the table within the **Boundary** tab, leaving only the fields for atom fractions (that is, **c**, **h**, **o**, **n**, and **s**).

15.4.4. Modeling Liquid Fuel Combustion Using the Non-Premixed Model

Liquid fuel combustion can be modeled with the discrete phase and non-premixed models. In ANSYS Fluent, the fuel vapor, which is produced by evaporation of the liquid fuel, is defined as the fuel stream. (See Defining the Stream Compositions (p. 1042).) The liquid fuel that evaporates within the domain appears as a source of the mean fuel mixture fraction.

Within ANSYS Fluent, you define the liquid fuel discrete-phase model in the usual way. The gas phase (oxidizer) flow inlet is modeled using an inlet mixture fraction of zero and the fuel droplets are introduced as discrete phase injections (see Setting Initial Conditions for the Discrete Phase (p. 1265)). The property inputs for the liquid fuel droplets are unaltered by the non-premixed model (see Setting Material Properties for the Discrete Phase (p. 1313)). Note that when you are requested to input the gas phase species destination for the evaporating liquid, you should specify the species that comprises the evaporating stream.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the fuel stream was defined as a mixture of components, you should select the largest of these components as the "evaporating species". ANSYS Fluent will ensure that the mass evaporated from the liquid droplet enters the gas phase as a source of the fuel mixture that you defined. The evaporating species you select here is used only to compute the diffusion controlled driving force in the evaporation rate.

15.4.5. Modeling Coal Combustion Using the Non-Premixed Model

If your model involves coal combustion, the fuel and secondary stream compositions can be input in one of several ways. You can use a single mixture fraction (fuel stream) to represent the coal, defining the fuel composition as a mixture of volatiles and char (solid carbon). Alternatively, you can use two mixture fractions (fuel and secondary streams), defining the volatiles and char separately. In two-mixture-fraction models for coal combustion, the fuel stream represents the char and the secondary stream represents volatiles. This section describes the modeling options and special input procedures for coal combustion models using the non-premixed approach.

There are three options for coal combustion:

- When coal is the only fuel in the system, you can model the coal using two mixture fractions, where the primary stream represents the char and the secondary stream represents the volatiles. Generally, the char stream composition is defined as 100% C(s). The volatile stream composition is defined by selecting appropriate species and setting their mole or mass fractions. Alternatively, you can use the empirical method (input of atom fractions) for defining these compositions.

  Important

  Using two mixture fractions to model coal combustion is more accurate than using one mixture fraction as the volatile and char streams are modeled separately. However, the two-mixture-fraction model incurs significant additional computational expense since the multi-dimensional PDF integrations are performed at run time.

- When coal is the only fuel in the system, you can choose to model the coal using a single mixture fraction (the fuel stream). When this approach is adopted, the fuel composition you define includes both volatile species and char. Char is typically represented by including C(s) in the species list. You can define the fuel stream composition by selecting appropriate species and setting their mole fractions, or by using the empirical method (input of atom fractions). Definition of the composition is described in detail below.

  Important

  Using a single mixture fraction for coal combustion is less accurate than using two mixture fractions. However, convergence in ANSYS Fluent should be substantially faster than the two-mixture-fraction model.

- When coal is used with another (gaseous or liquid) fuel of different composition, you must model the coal with one mixture fraction and use a second mixture fraction to represent the second (gaseous or liquid) fuel. The stream associated with the coal composition is defined as detailed below for single-mixture-fraction models.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.4.5.1. Defining the Coal Composition: Single-Mixture-Fraction Models

When coal is modeled using a single mixture fraction (the fuel stream), the fuel stream composition can be input using the conventional approach or the empirical fuel approach.

- Conventional approach:

  To use the conventional approach, you will need to define the mixture of species in the coal and their mole or mass fractions in the fuel stream. Use the **Boundary** tab in the **Species Model** dialog box to input the list of species (for example, C_3H_8, CH_4, CO, CO_2, C(s)) that approximate the coal composition, and their mole or mass fractions.

  Note that C(s) is used to represent the char content of the coal. For example, consider a coal that has a molar composition of 40% volatiles and 60% char on a dry ash free (DAF) basis. Assuming the volatiles can be represented by an equimolar mixture of C_3H_8 and CO, the fuel stream composition defined in the **Boundary** tab would be C_3H_8=0.2, CO = 0.2, and C(s)=0.60. Note that the coal composition should always be defined on an ash-free basis, even if ash will be considered in the ANSYS Fluent calculation.

  To define ash properties, go to the **Create/Edit Materials** dialog box and select **combusting-particle** as the **Material Type**.

  The following table illustrates the conversion from a typical mass-based proximate analysis to the species fraction inputs required by ANSYS Fluent. Note that the conversion requires that you make an assumption regarding the species representing the volatiles. Here, the volatiles are assumed to exist as an equimolar mix of propane and carbon monoxide.

Proximate Analysis	Weight %	Mass Fraction (DAF)	Moles (DAF)	Mole Fraction (DAF)
Volatiles	30			
– C_3H_8		0.2035	0.004625	0.07134
– CO		0.1295	0.004625	0.07134
Fixed Carbon (C(s))	60	0.667	0.05558	0.85732
Ash	10	-	-	-
(Total)			0.06483	1.0

  Moisture in the coal can be considered by adding it in the fuel composition as liquid water, H_2O(l). The moisture can also be defined as water vapor, H_2O, provided that the corresponding latent heat is included in the discrete phase material inputs in ANSYS Fluent. If the liquid water is used as a boundary species, it should be removed from the list of excluded species (see Forcing the Exclusion and Inclusion of Equilibrium Species (p. 1053)).

  Important

  Note that if water is included in the coal, the water release is not modeled as evaporation, which is typically the case in the wet combustion model, described in Particle Types (p. 1268).

- Empirical fuel approach:

To use the empirical approach, enable the **Empirical Fuel Stream** option in the **Chemistry** tab. This method is ideal if you have an elemental analysis of the coal.

In the **Chemistry** tab, enter the lower heating value and mean specific heat of the coal. ANSYS Fluent will use these inputs to determine the mole fractions of the chemical species you have included in the system. Then, in the **Boundary** tab, define the atom fractions of C, H, N, S, and O in the fuel stream.

Note that for both of these composition input methods, you should take care to distinguish atomic carbon, C, from solid carbon, C(s). Atomic carbon should only be selected if you are using the empirical fuel input method.

See Additional Coal Modeling Inputs in ANSYS Fluent (p. 1049) for details about further inputs for modeling coal combustion.

15.4.5.2. Defining the Coal Composition: Two-Mixture-Fraction Models

You can model coal using the two mixture fractions model, where the primary stream represents the char and the secondary stream represents the volatiles.

As in single-mixture-fraction cases, the fuel stream and secondary stream compositions in a two-mixture-fraction case can be entered using either the conventional approach or the empirical fuel approach.

- Conventional approach:

  To use the conventional approach, you will need to define the mixture of species in the coal and their mole or mass fractions in the fuel and secondary streams.

  Use the **Boundary** tab of **Species Model** dialog box to define the mole or mass fractions of volatile species in the secondary stream (for example, C_3H_8, CH_4, CO, CO_2, C(s)). Next, define the mole or mass fractions of species used to represent the char. Generally, you will specify 100% C(s) for the fuel stream.

- Empirical fuel approach:

  To use the empirical fuel approach, enable the **Empirical Secondary Stream** option in the **Chemistry** tab for the volatile (secondary) stream. This method is ideal if you have an elemental analysis of the coal.

  In the **Chemistry** tab, input the lower heating value and mean specific heat of the coal volatiles. Then, in the **Boundary** tab, define the mole or mass fractions of species used to represent the char. Generally, you will specify 100% C(s) for the fuel stream. Finally, define the atom fractions of C, H, N, S, and O in the volatiles. ANSYS Fluent will use these inputs to determine the mole fractions of the chemical species you have included in the system. For example, consider coal with the following DAF (dry ash free) data and elemental analysis:

Proximate Analysis	Wt % (dry)	Wt % (DAF)
Volatiles	28	30.4
Char (C(s))	64	69.6
Ash	8	-

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Element	Wt % (DAF)	Wt % (DAF)
C	89.3	89.3
H	5.0	5.0
O	3.4	3.4
N	1.5	2.3
S	0.8	-

(Note that in the final column, for modeling simplicity, the sulfur content of the coal has been combined into the nitrogen mass fraction.)

You can combine the proximate and ultimate analysis data to yield the following elemental composition of the volatile stream:

Element	Mass	Mass Fraction	Moles	Mole Fraction
C	(89.3 - 69.6)	0.65	5.4	0.24
H	5.0	0.16	16	0.70
O	3.4	0.11	0.7	0.03
N	2.3	0.08	0.6	0.03
Total	30.4		22.7	

This adjusted composition is used to define the secondary stream (volatile) composition.

The lower heating value of the volatiles can be computed from the known heating value of the coal and the char (DAF):

$LCV_{coal,DAF}=35.3\,\mathrm{MJ/kg}$

$LCV_{char,DAF}=32.9\,\mathrm{MJ/kg}$

You can compute the heating value of the volatiles as

$$LCV_{vol}=\frac{35.3\,\mathrm{MJ/kg}-0.696\times 32.9\,\mathrm{MJ/kg}}{0.304} \tag{15.2}$$

or

$$LCV_{vol}=40.795\,\mathrm{MJ/kg} \tag{15.3}$$

Note that for both of these composition input methods, you should take care to distinguish atomic carbon, C, from solid carbon, C(s). Atomic carbon should only be selected if you are using the empirical fuel input method.

15.4.5.3. Additional Coal Modeling Inputs in ANSYS Fluent

Within ANSYS Fluent, the DPM coal combustion simulation is defined as usual when the non-premixed combustion model is selected. The air (oxidizer) inlets are defined as having a mixture fraction value of zero. No gas phase fuel inlets will be included and the sole source of fuel will come from the coal devolatilization and char burnout. The coal particles are defined as injections using the **Set Injection Properties** dialog box in the usual way, and physical properties for the coal material are specified as described in Setting Material Properties for the Discrete Phase (p. 1313). Remember the following issues when you are defining injections and discrete-phase material properties for coal materials:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- In the **Set Injection Properties** dialog box, you will specify for the **Oxidizing Species** one of the components of the oxidizer stream. This species concentration field will be used to calculate the diffusion-controlled driving force in the char burnout law (if applicable), and is O_2 by default.

  The specification of the char and volatile streams differs depending on the type of model you are defining:

  – If coal is modeled using a single mixture fraction, the gas phase species representing the volatiles and the char combustion are represented by the mixture fraction used by the non-premixed combustion model.

  – If coal is modeled using two mixture fractions, rather than specifying a destination species for the volatiles and char, you will instead specify the **Devolatilizing Stream** (as secondary) and the **Char Stream** (as primary).

  – If coal is modeled using one mixture fraction, and another fuel is modeled using a second mixture fraction, you should specify the stream representing the coal as both the **Devolatilizing Stream** and the **Char Stream**.

- In the **Create/Edit Materials** dialog box, **Vaporization Temperature** should be set equal to the fuel inlet temperature. This temperature controls the onset of the devolatilization process. The fuel inlet temperature that you define in the **Boundary** tab of the **Species Model** dialog box should be set to the temperature at which you want to initiate devolatilization. This way, the look-up tables will include the appropriate temperature range for your process.

- In the **Create/Edit Materials** dialog box, **Volatile Component Fraction** and **Combustible Fraction** should be set to values that are consistent with the coal composition used to define the fuel (and secondary) stream composition.

- Also in the **Create/Edit Materials** dialog box, you will be prompted for the **Burnout Stoichiometric Ratio** and for the **Latent Heat**. The **Burnout Stoichiometric Ratio** is used in the calculation of the diffusion controlled burnout rate but has no other impact on the system chemistry when the non-premixed combustion model is used. The **Burnout Stoichiometric Ratio** is the mass of oxidant required per mass of char. The default value of 2.67 assumes that C(s) is oxidized by O_2 to yield CO_2. The **Latent Heat** input determines the heat required to generate the vapor phase volatiles defined in the non-premixed system chemistry. You can usually set this value to zero when the non-premixed model is used, since your definition of volatile species will have been based on the overall heating value of the coal. However, if the coal composition includes the water content, the latent heat should be set as follows:

  – Set the latent heat to zero if the water content of the coal has been defined as **H2O(L)**. In this case, the system chemistry will include the latent heat required to vaporize the liquid water.

  – Set the latent heat to the value for water (2.25×10^6 J/kg), adjusted by the mass loading of water in the volatiles, if the water content of the coal has been defined using water vapor, **H2O**. In this case, the water content you defined will be evolved along with the other species in the coal but the system chemistry does not include the latent heat effect.

- The **Density** you define for the coal in the **Create/Edit Materials** dialog box should be the apparent density, including ash content.

- You will not be asked to define the **Heat of Reaction for Burnout** for the char combustion.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

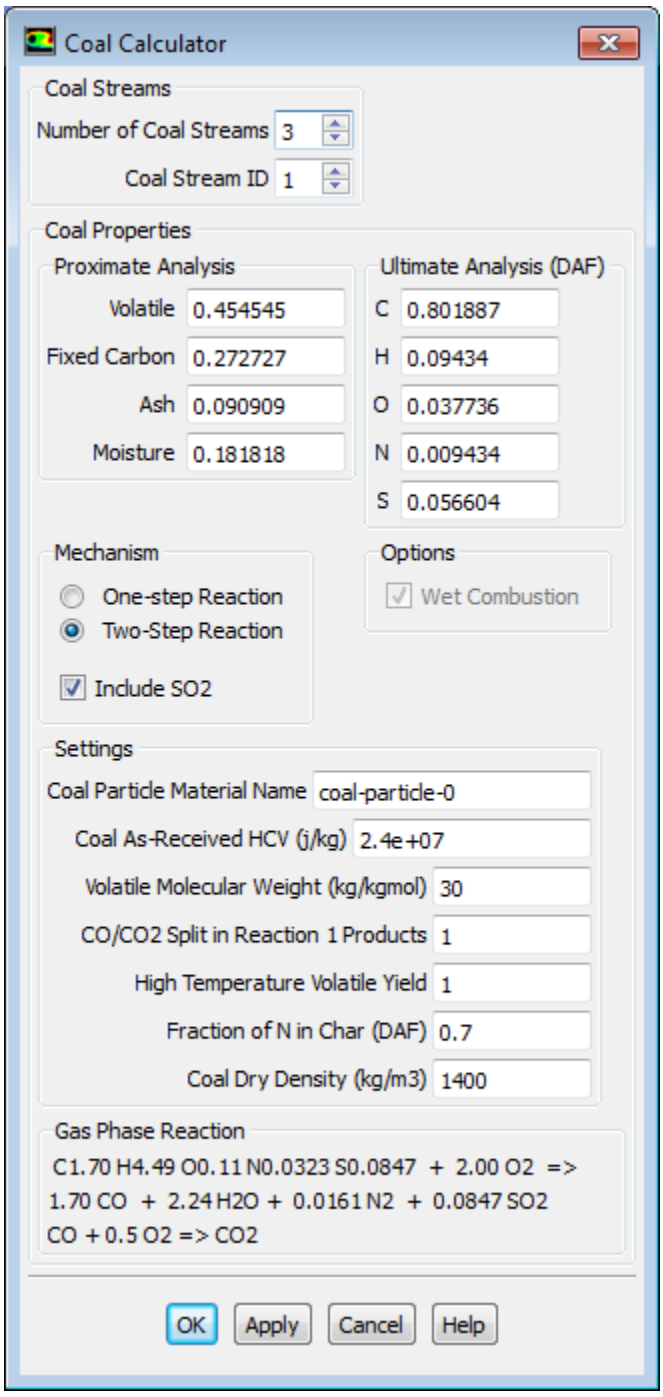
15.4.5.4. Postprocessing Non-Premixed Models of Coal Combustion

ANSYS Fluent reports the rate of volatile release from the coal using the **DPM Evaporation/Devolatilization** postprocessing variable. The rate of char burnout is reported in the **DPM Burnout** variable.

15.4.5.5. The Coal Calculator

The **Coal Calculator** dialog box automates the calculations described above for setting up a coal case from the proximate and ultimate analyses.

Figure 15.12: The Coal Calculator Dialog Box



The inputs to the **Coal Calculator** dialog box are:

1. Coal **Proximate Analysis**, which is the mass fraction of **Volatile, Fixed Carbon, Ash** and **Moisture** in the coal.

2. Coal **Ultimate Analysis**, which is the mass fraction of atomic **C, H, O, N** and optionally **S**, in the Dry-Ash-Free (DAF) coal.

3. The option to use a **Secondary Stream**. If enabled, the two mixture fraction model will be set with the primary stream representing char as $C<s>$, and an empirical secondary stream representing the volatiles.

4. The **Coal Particle Material Name**. A DPM Combusting Particle Material will be created with this name. The default name is *coal-particle*.

5. The **Coal As-Received HCV** (Higher Calorific Value).

6. The **High Temperature Volatile Yield**. Proximate analyses are generally done with slower heating rates and lower temperatures than would occur in a real flame. Therefore, enhanced devolatization at higher temperatures can cause the volatile yield to exceed the proximate analysis fraction. To model this, the actual volatile fraction used is calculated as that specified in the **Proximate Analysis** input multiplied by the **High Temperature Volatile Yield**. The actual **Fixed Carbon** fraction is then calculated as one minus the sum of the actual **Volatile**, **Ash**, and **Moisture** fractions.

7. **Fraction of N in Char (DAF)**. This input is used in calculating the split of atomic nitrogen for the Fuel NOx model.

   When the **OK** button is clicked, ANSYS Fluent makes the following changes:

   a. The empirical fuel atomic compositions in the **Boundary** tab are set, and the **Non-Adiabatic** model is enabled as required for DPM. The Empirical Fuel Lower Calorific Value is calculated as follows. First the DAF LCV of the coal is computed as,

   $$LCV_{coal}^{DAF} = \frac{HCV_{coal}^{ar} - h_{H_2O}^{latent} Moisture}{1 - Moisture - Ash} - \frac{H_{ar} W_{H_2O}}{2 W_H} h_{H_2O}^{latent} \tag{15.4}$$

   where $Moisture$ and Ash are the proximate moisture and ash fractions, H_{ar} is the ultimate H fraction, W_{H_2O} and W_H are the molecular weight of water and atomic hydrogen, respectively, and $h_{H_2O}^{latent}$ is the latent heat of water.

   If you are using the **Secondary Stream** option, LCV_{vol} is calculated from LCV_{coal}^{DAF} using,

   $$LCV_{vol} = \frac{LCV_{coal}^{DAF}\left(1 - Moisture - Ash\right) - LCV_{char} FixedCarbon}{Volatile} \tag{15.5}$$

   where $FixedCarbon$ and $Volatile$ are the proximate fixed carbon and volatile fractions, respectively.

   b. A combusting particle material is created with **Volatile Component Fraction** and **Combustible Fraction** calculated from the ultimate and proximate analyses. The Discrete Phase Model (DPM) is enabled.

   c. For the Fuel NOx model, the char N conversion is set to *NO*, and the Fuel NOx **Volatile** and **Char** mass fractions are set according to the ultimate and proximate compositions. Note that even though some of the Fuel NOx parameters are changed, the Fuel NOx model itself is not enabled.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After the **Coal Calculator** has set up the relevant models, you must build the PDF Table by clicking **Calculate PDF Table** in the **Table** tab. You will also need to create injections if you have not done this yet. After converging your coal combustion case, you may want to enable the NOx model for postprocessing nitrogen-oxide pollutants.

15.5. Setting Up Control Parameters

For information about setting up control parameters, see the following sections:

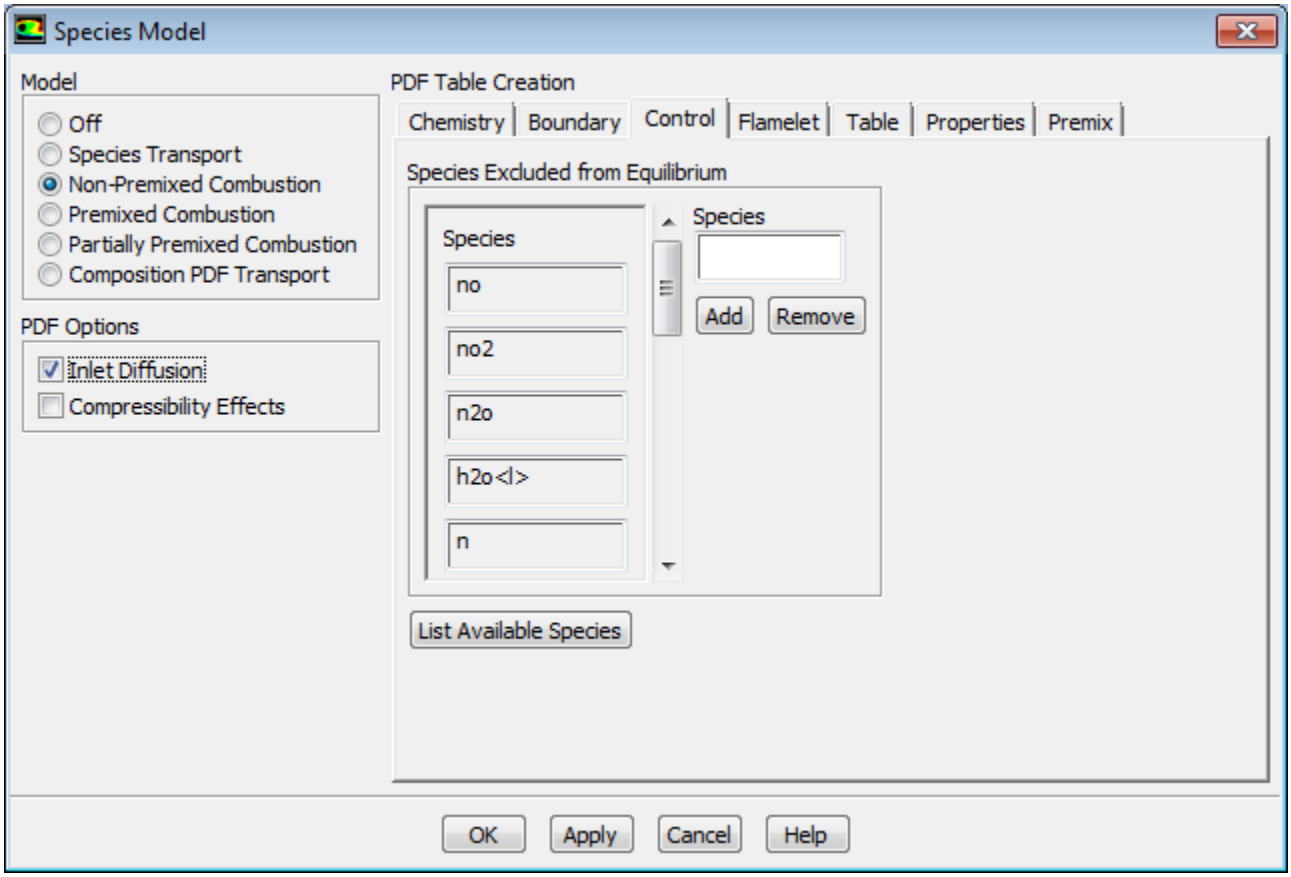
15.5.1. Forcing the Exclusion and Inclusion of Equilibrium Species
15.5.2. Defining the Flamelet Controls
15.5.3. Zeroing Species in the Initial Unsteady Flamelet

15.5.1. Forcing the Exclusion and Inclusion of Equilibrium Species

Because ANSYS Fluent calculates all intermediate and product species automatically during the equilibrium calculation, certain species will be included that are generally not in chemical equilibrium. Principal among these are the NOx species. Specifically, the NOx reaction rates are slow and should not be treated using an equilibrium assumption. Instead, the NOx concentration is predicted most accurately using the ANSYS Fluent NOx postprocessor, where finite-rate kinetics are included (see NOx Formation (p. 1153)). The NOx species can be safely excluded from the equilibrium calculation since they are present at low concentrations and have little impact on the density, temperature, and other species.

To force the exclusion of a species from the equilibrium calculation, click the **Control** tab in the **Species Model** dialog box (Figure 15.13: The Species Model Dialog Box (Control Tab) (p. 1053)).

Figure 15.13: The Species Model Dialog Box (Control Tab)



Under **Species Excluded From Equilibrium**, enter the chemical formula for the desired species in the **Add/Remove Species** field. Next, click **Add** to add the species to the **Species** list or **Remove** to remove an existing species from the **Species** list.

If there are species that you want to include in your PDF table that would be ignored by ANSYS Fluent due to their low concentration (for example, CH, CH_2, CH_3 for the NOx calculation), you can force ANSYS Fluent to include them using the text interface:

`define` → `models` → `species` → `non-premixed-combustion`

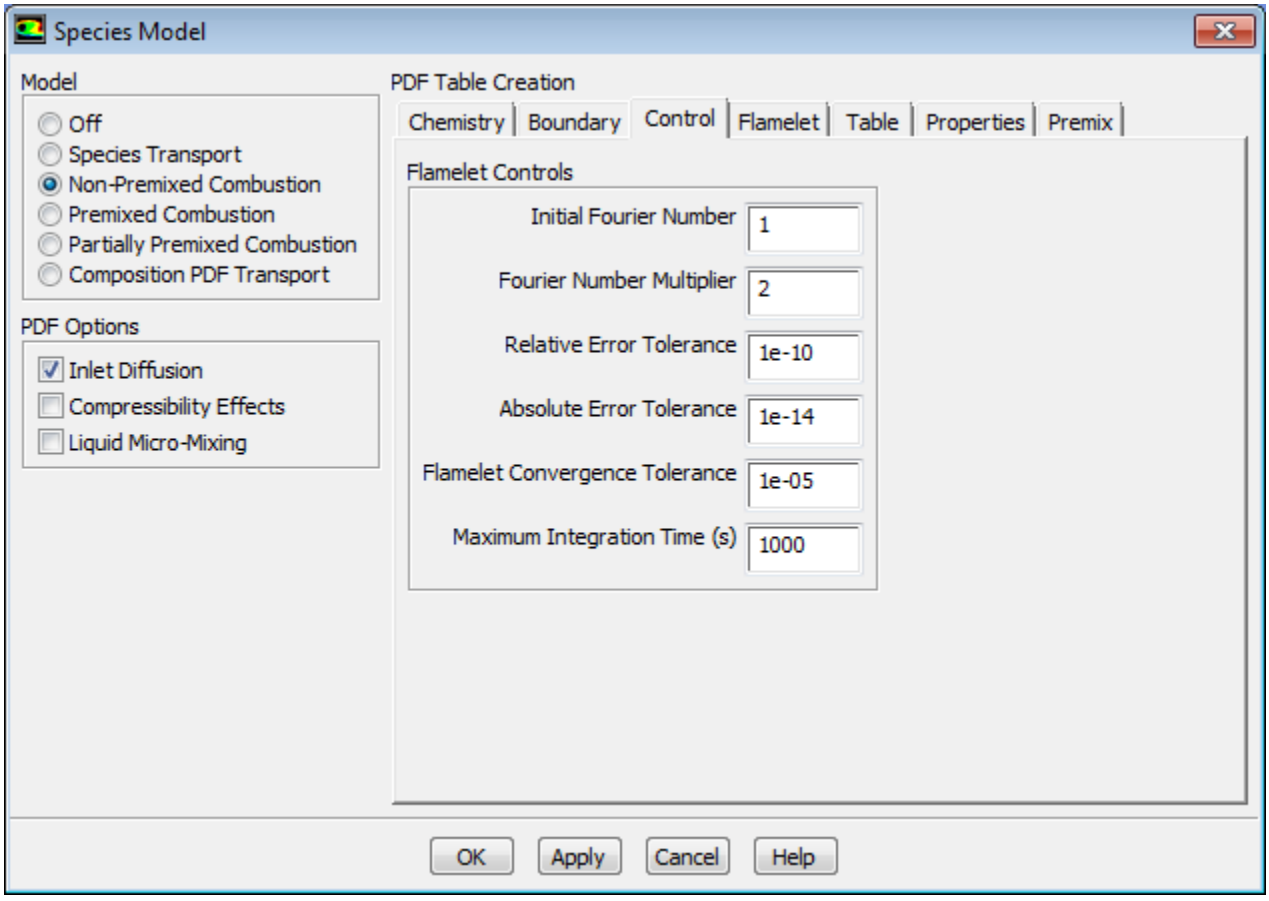
When the console window prompts you with `Force Equilibrium Species to Include...`, specify the appropriate species by entering the chemical formula(s) in double quotes (for example, `"ch"`, `"ch2"`).

Note that you will have to first set up the inputs for the fuel and oxidizer before you are given the option to include the species.

15.5.2. Defining the Flamelet Controls

When the steady diffusion flamelet model is selected, and you have created or imported a flamelet, you can adjust the controls for the flamelet solution in the **Control** tab of the **Species Model** dialog box (Figure 15.14: The Species Model Dialog Box (Control Tab) for the Steady Diffusion Flamelet Model (p. 1054)).

Figure 15.14: The Species Model Dialog Box (Control Tab) for the Steady Diffusion Flamelet Model



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Initial Fourier Number** sets the first time step for the solution of the flamelet equations (Equation 8.46 and Equation 8.47 in the Theory Guide). This first time step is calculated as the explicit stability-limited diffusion time step multiplied by this value. If the solution diverges before the first time step is complete, the value should be lowered.

The **Fourier Number Multiplier** increases the time step at subsequent times. Every time step after the first is multiplied by this value. If the solution diverges after the first time step, this value should be reduced.

During the numerical integration of the flamelet equations, the local error is controlled to be less than

$$error_{loc,i}=\varepsilon_{rel}\phi_i+\varepsilon_{abs} \tag{15.6}$$

where ϕ_i represents the species mass fractions and temperature at point i in the 1D flamelet. ε_{rel} is the value of the **Relative Error Tolerance** and ε_{abs} is the value of the **Absolute Error Tolerance**, both of which you can specify.

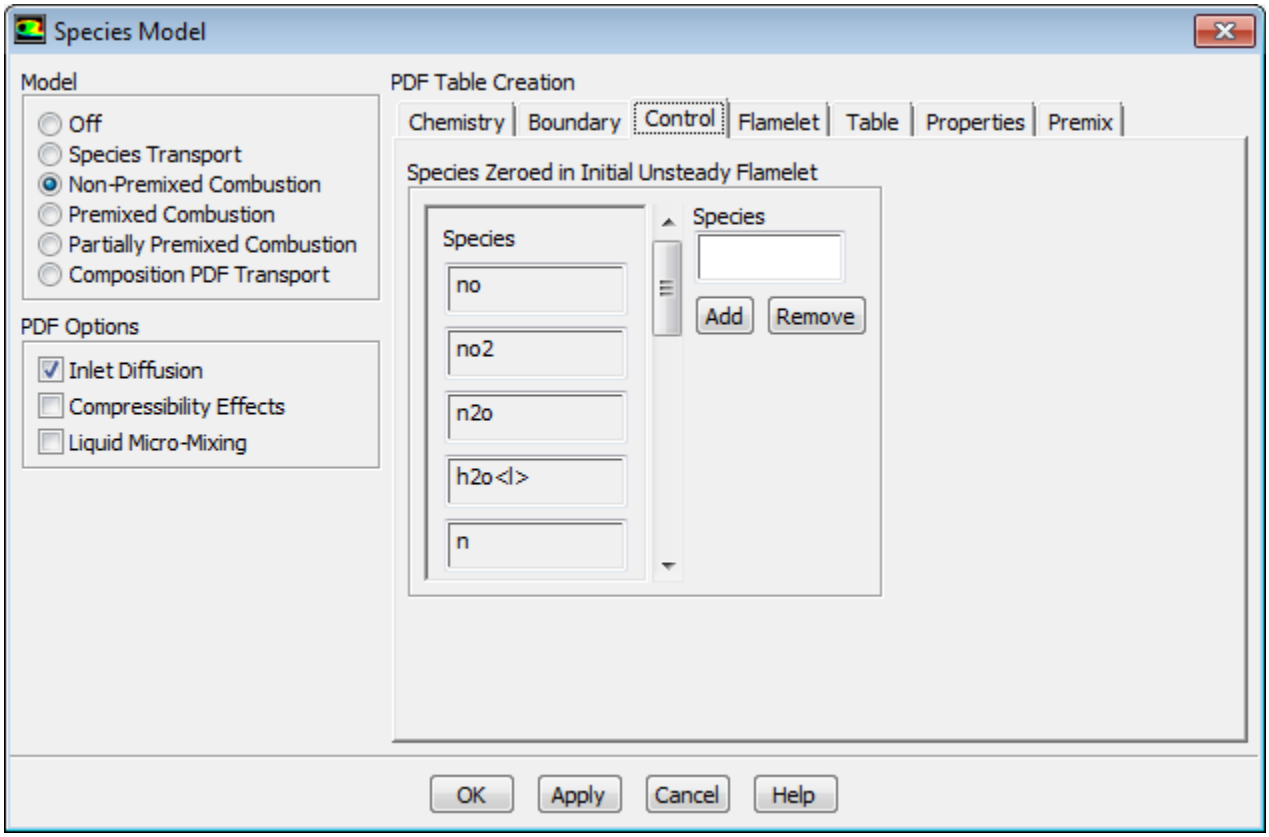
Because steady flamelets are obtained by time-stepping, they are considered converged only when the maximum absolute change in species fraction or temperature at any discrete mixture-fraction point is less than the specified **Flamelet Convergence Tolerance**. Between time steps, the flamelet species fractions and temperature will sometimes oscillate, which causes absolute changes that are always greater than the flamelet convergence tolerance. In such cases, ANSYS Fluent will stop the flamelet calculation after the total elapsed time has exceeded the **Maximum Integration Time**.

15.5.3. Zeroing Species in the Initial Unsteady Flamelet

When modeling gas-phase combustion using the Eulerian unsteady laminar flamelet model, the flamelet fields are initialized to a burning, steady-flamelet solution in order to model ignition. However, assuming steady-flamelet profiles for slow-forming species is inaccurate. A better approximation is to identify the slow species and to set them to zero, which is done in the **Control** tab. By default, ANSYS Fluent selects some NOx species (NO, NO_2, N_2O, N, NH, NH_2, NH_3, NNH, HCN, HNO, CN, $H2CN$, $HCNN$, $HCNO$, $HOCN$, $HNCO$, HCO), as well as liquid water H_2O<l> and solid carbon C<s> to be zeroed. See Figure 15.15: Method to Zero Out the Slow Chemistry Species (p. 1056).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.15: Method to Zero Out the Slow Chemistry Species



15.6. Calculating the Flamelets

For information about calculating flamelets, see the following sections:

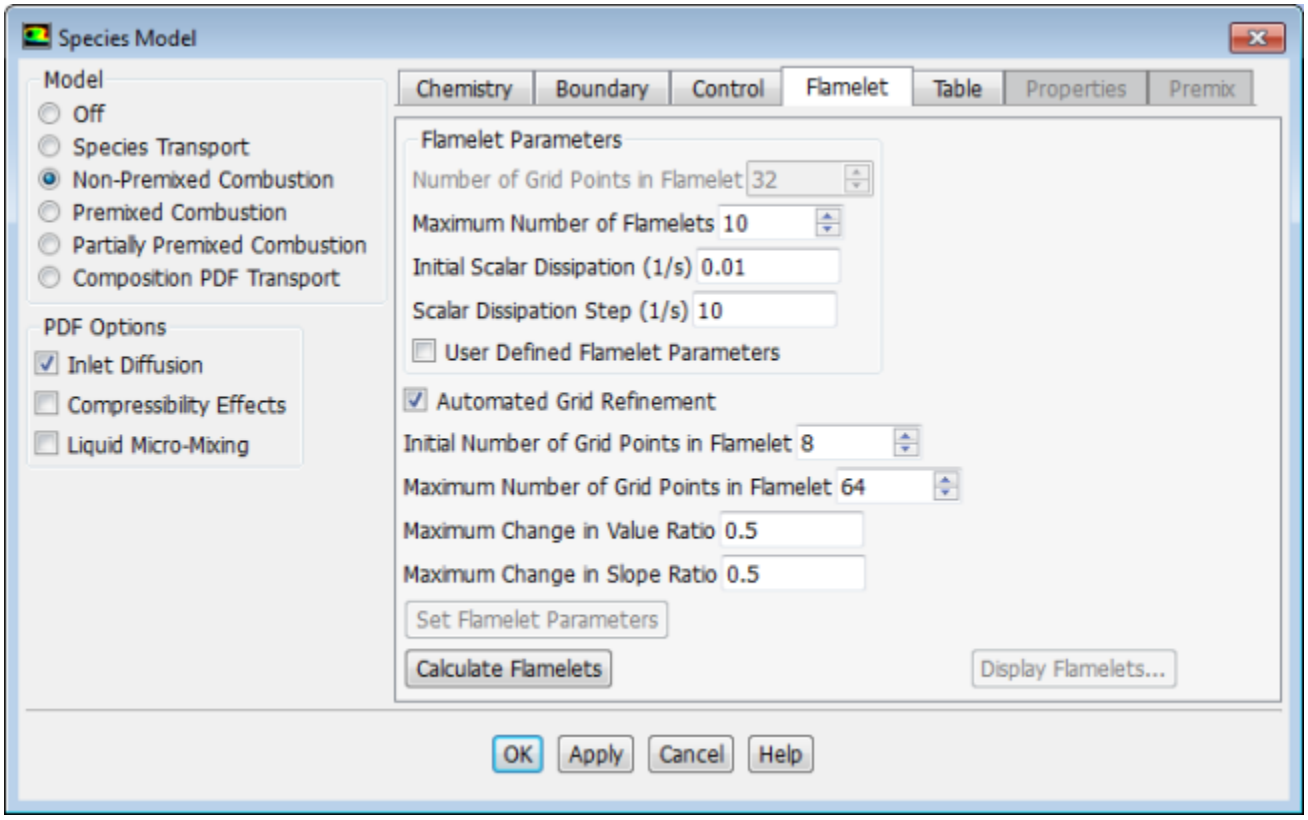
15.6.1. Steady Diffusion Flamelet
15.6.2. Unsteady Diffusion Flamelet
15.6.3. Saving the Flamelet Data
15.6.4. Postprocessing the Flamelet Data

15.6.1. Steady Diffusion Flamelet

In the **Flamelet** tab of the **Species Model** dialog box (Figure 15.16: The Species Model Dialog Box (Flamelet Tab) (p. 1057)), you will enter values for parameters of the flamelet(s).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.16: The Species Model Dialog Box (Flamelet Tab)



The **Flamelet Parameters** are as follows:

Number of Grid Points in Flamelet

specifies the number of mixture fraction grid points distributed between the oxidizer ($f=0$) and the fuel ($f=1$). Increased resolution will provide greater accuracy, but since the flamelet species and temperature are solved coupled and implicit in f space, the solution time and memory requirements increase greatly with the number of f grid points.

Maximum Number of Flamelets

specifies the maximum number of flamelet profiles to be calculated. If the flamelet extinguishes before this number is reached, flamelet generation is halted and the actual number of flamelets in the flamelet library will be less than this value.

Initial Scalar Dissipation

is the scalar dissipation of the first flamelet in the library. This corresponds to χ_0 in Equation 8.52 in the Theory Guide.

Scalar Dissipation Step

specifies the interval between scalar dissipation values (in s^{-1}) for which multiple flamelets will be calculated. This corresponds to $\Delta\chi$ in Equation 8.52 in the Theory Guide.

User Defined Flamelet Parameters

enables you to hook a user-defined function for scalar dissipation and mean mixture fraction (or progress variable) grid discretization

Automated Grid Refinement employs an adaptive algorithm, which inserts grid points so that the change of values, as well as the change of slopes, between successive grid points is less than user-

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

specified tolerances. For information about this option, refer to Steady Diffusion Flamelet Automated Grid Refinement in the Theory Guide.

Initial Number of Grid Points in Flamelet
calculates a steady solution on a coarse grid, with a default of 8. See Equation 8.53 in the Theory Guide.

Maximum Number of Grid Points in Flamelet
has a default of 64.

Maximum Change in Value Ratio
has a default of 0.5 and is ε_v in Equation 8.27 in the Theory Guide.

Maximum Change in Slope Ratio
has a default of 0.5 and is ε_s in Equation 8.28 in the Theory Guide.

Click **Calculate Flamelets** to begin the diffusion flamelet calculation. Sample output for a flamelet calculation is shown below.

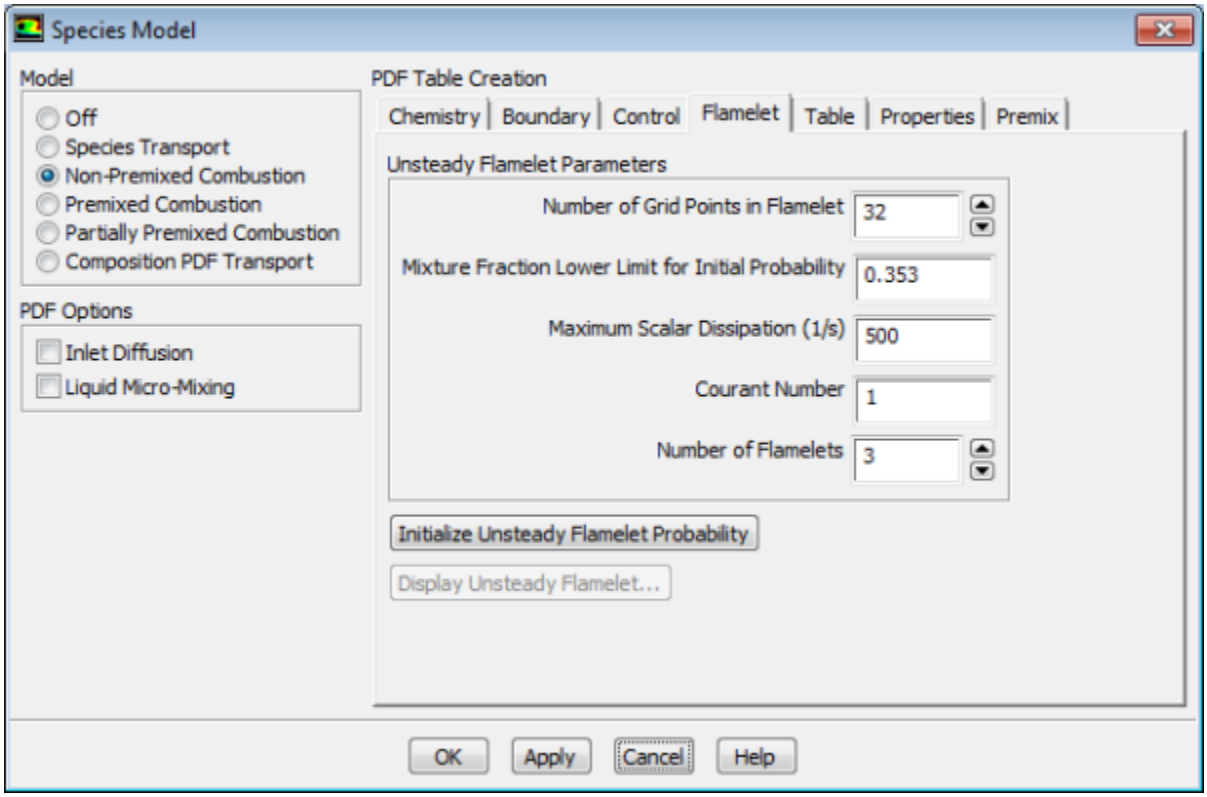
```
 Generating flamelet 1 at scalar dissipation  0.01 /s

Time (s)   Temp (K)   Residual
1.679e-05   2233.7   3.779e+00
5.038e-05   2233.0   7.734e-02
1.175e-04   2231.5   1.648e-01
2.519e-04   2228.6   3.652e-01
5.206e-04   2223.6   8.295e-01
1.058e-03   2215.7   2.100e+00
2.133e-03   2205.5   3.540e+00
4.282e-03   2197.0   4.607e+00
8.581e-03   2193.6   6.639e+00
1.718e-02   2193.1   4.905e+00
3.437e-02   2193.4   5.792e+00
6.877e-02   2194.3   4.659e+00
1.375e-01   2195.3   3.922e+00
2.751e-01   2192.2   3.181e+00
5.502e-01   2188.6   2.549e+00
1.100e+00   2184.8   1.639e+00
2.201e+00   2182.9   4.604e+00
4.402e+00   2186.8   1.307e+00
8.804e+00   2189.6   4.420e-01
1.761e+01   2190.0   8.581e-02
3.522e+01   2190.0   1.199e-02
7.043e+01   2190.0   1.735e-03
1.409e+02   2190.0   4.217e-04
2.817e+02   190.0    6.892e-05
5.635e+02   2190.0   6.777e-06

Flamelet successfully generated
```

15.6.2. Unsteady Diffusion Flamelet

In the **Flamelet** tab of the **Species Model** dialog box (Figure 15.17: The Flamelet Tab for the Unsteady Diffusion Flamelet Model (p. 1059)), you will enter values for parameters of the flamelet.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.17: The Flamelet Tab for the Unsteady Diffusion Flamelet Model



The **Unsteady Flamelet Parameters** are as follows:

Number of Grid Points in Flamelet
specifies the number of mixture fraction grid points distributed between the oxidizer ($f=0$) and the fuel ($f=1$). Increased resolution will provide greater accuracy, but since the flamelet species and temperature are solved coupled and implicit in f space, the solution time and memory requirements increase with the number of f grid points.

Mixture Fraction Lower Limit for Initial Probability
is the mixture fraction above which the marker probability will be initialized to 1, and below which the marker probability will be initialized to 0. In general, it should be set greater than the stoichiometric mixture fraction.

Maximum Scalar Dissipation
is where flamelets extinguish at large scalar dissipation (mixing) rates. To prevent excessive mixing in the flamelet, ANSYS Fluent allows you to specify a **Maximum Scalar Dissipation** rate for the 1D flamelet equations. A reasonable value for this is the steady flamelet extinction scalar dissipation. The default value of 30/s is near the steady extinction scalar dissipation of a methane-air flame at standard temperature and pressure.

Courant Number
is the number at which ANSYS Fluent automatically selects the time step for the probability equation based on this convective Courant number.

Number of Flamelets
is the number of unsteady laminar flamelets to be initiated in the simulation. The probability marker equation will be solved for each flamelet.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Click **Initialize Unsteady Flamelet Probability** to initialize the unsteady flamelet and its probability marker equation. ANSYS Fluent is now ready for postprocessing the 1D unsteady flamelet and the 2D/3D unsteady marker probability equation.

15.6.3. Saving the Flamelet Data

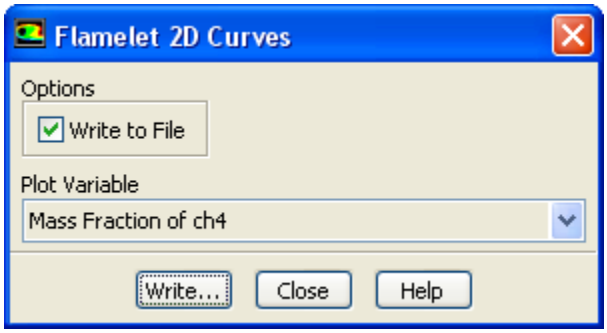
The flamelet tables may be written to file for import into later sessions of ANSYS Fluent. You may want to do this, for example, to change the number of discretization points in the PDF table, or to plot the flamelet profiles in ANSYS Fluent. The flamelet tables should be saved before you create the PDF table:

File → **Write** → **Flamelet...**

15.6.4. Postprocessing the Flamelet Data

For the flamelet model, you can display or write flamelet curves. Click the **Display Flamelets...** or **Display Unsteady Flamelet...** button. If you have a single flamelet, as for the unsteady diffusion flamelet model, you can access the **Flamelet 2D Curves** dialog box (Figure 15.18: The Flamelet 2D curves Dialog Box (p. 1060)).

Figure 15.18: The Flamelet 2D curves Dialog Box

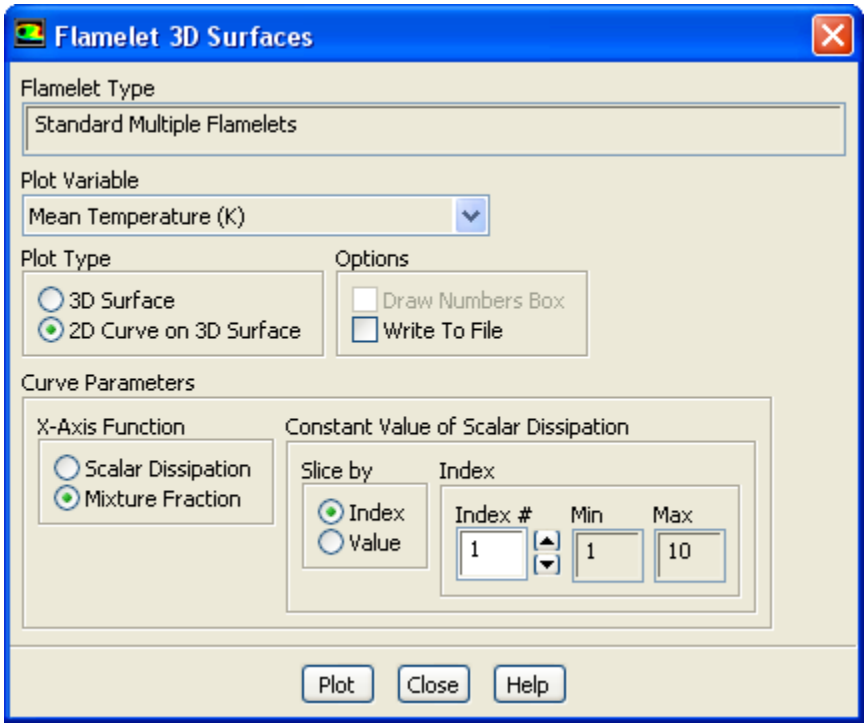


For the steady diffusion flamelet model with more than one flamelet, you can display 2D plots and 3D surfaces showing the variation of species fraction or temperature with the mean mixture fraction or scalar dissipation using the **Flamelet 3D Surfaces** dialog box (for example, Figure 15.19: The Flamelet 3D Surfaces Dialog Box (p. 1061)).

To access this dialog box, click the **Display Flamelets...** button in the **Flamelet** tab of the **Species Model** dialog box, as shown in Figure 15.16: The Species Model Dialog Box (Flamelet Tab) (p. 1057).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.19: The Flamelet 3D Surfaces Dialog Box



To display the flamelet tables graphically, use the following procedure:

1. In the **Flamelet 3D Surfaces** dialog box, from the **Plot Variable** drop-down list, select the variable you want to display graphically.

2. Specify the **Plot Type** as either **3D Surface** or **2D Curve on 3D Surface**.

   - For a 3D surface, enable or disable **Draw Numbers Box** under **Options**. When this option is turned on, the display will include a wireframe box with the numerical limits in each coordinate direction.

   - For a 2D curve on a 3D surface:

     a. Specify whether you want to write the plot data to a file by toggling **Write To File** under **Options**.

     b. Specify the **X-Axis Function** against which the plot variable will be displayed by selecting **Scalar Dissipation** (χ), or **Mixture Fraction** (f). The variable that is not selected will be held constant.

     c. Specify the type of discretization (that is, how the flamelet data will be sliced) for the variable that is being held constant (under **Constant Value of Mixture Fraction** or **Constant Value of Scalar Dissipation**).

        – If you selected **Index** under **Slice by**, specify the discretization **Index** of the variable that is being held constant. The range of integer values that you are allowed to choose from is displayed under **Min** and **Max**, and is equivalent to the number of points specified for that variable in the **Flamelet** tab of the **Species Model** dialog box (see Calculating the Flamelets (p. 1056)).

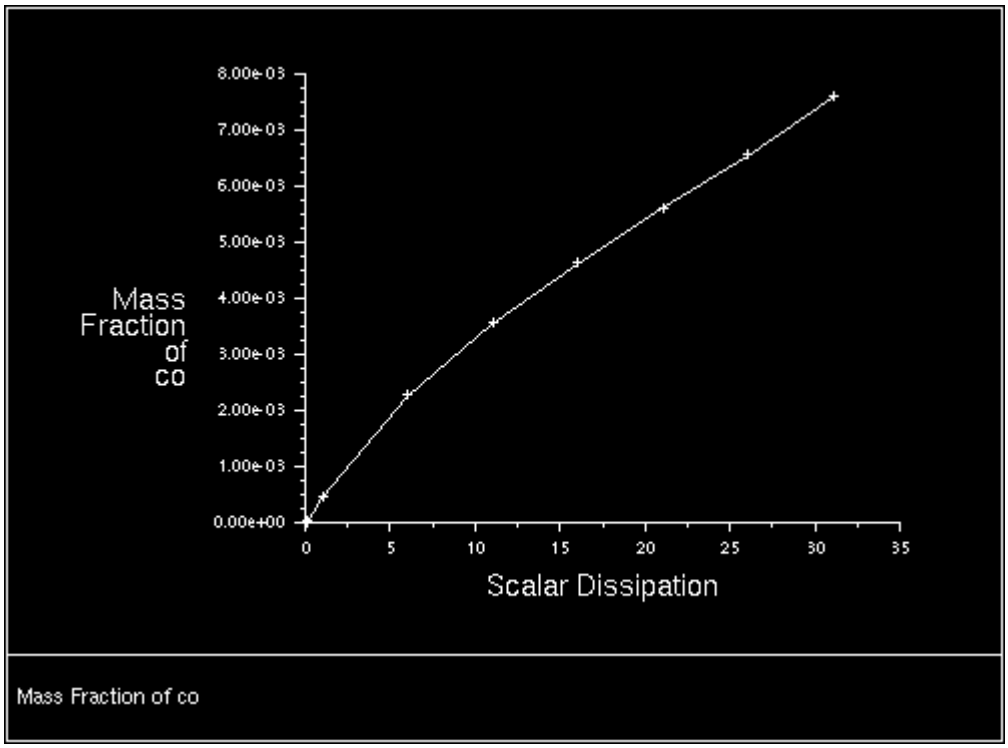
        – If you selected **Value** under **Slice by**, specify the numerical **Value** of the variable that is being held constant. The range of values that you can specify is displayed under **Min** and **Max**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Write or display the flamelet table results. If you have turned on the **Write To File** option for a 2D plot, click **Write** and specify a name for the file in The Select File Dialog Box (p. 19). Otherwise, click **Plot** or **Display** as appropriate to display a 2D plot or 3D surface in the graphics window.

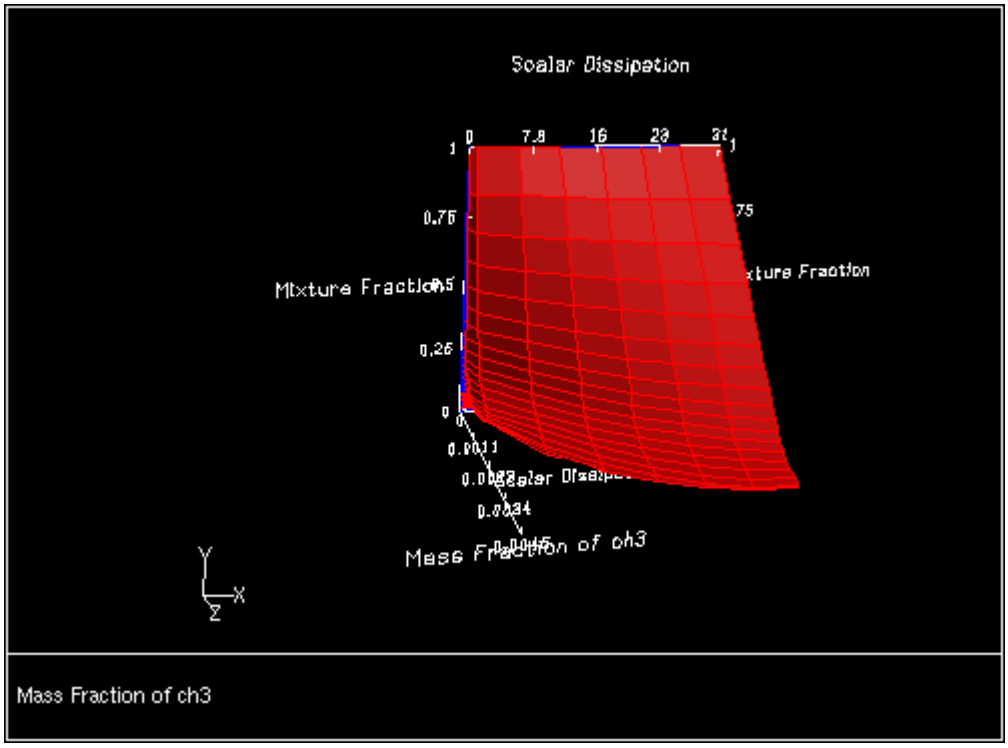
Figure 15.20: Example 2D Plot of Flamelet Data (p. 1062) and Figure 15.21: Example 3D Plot of Flamelet Data (p. 1063) show examples of a 2D curve plot and 3D surface plot of a flamelet table.

Figure 15.20: Example 2D Plot of Flamelet Data



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.21: Example 3D Plot of Flamelet Data



15.7. Calculating the Look-Up Tables

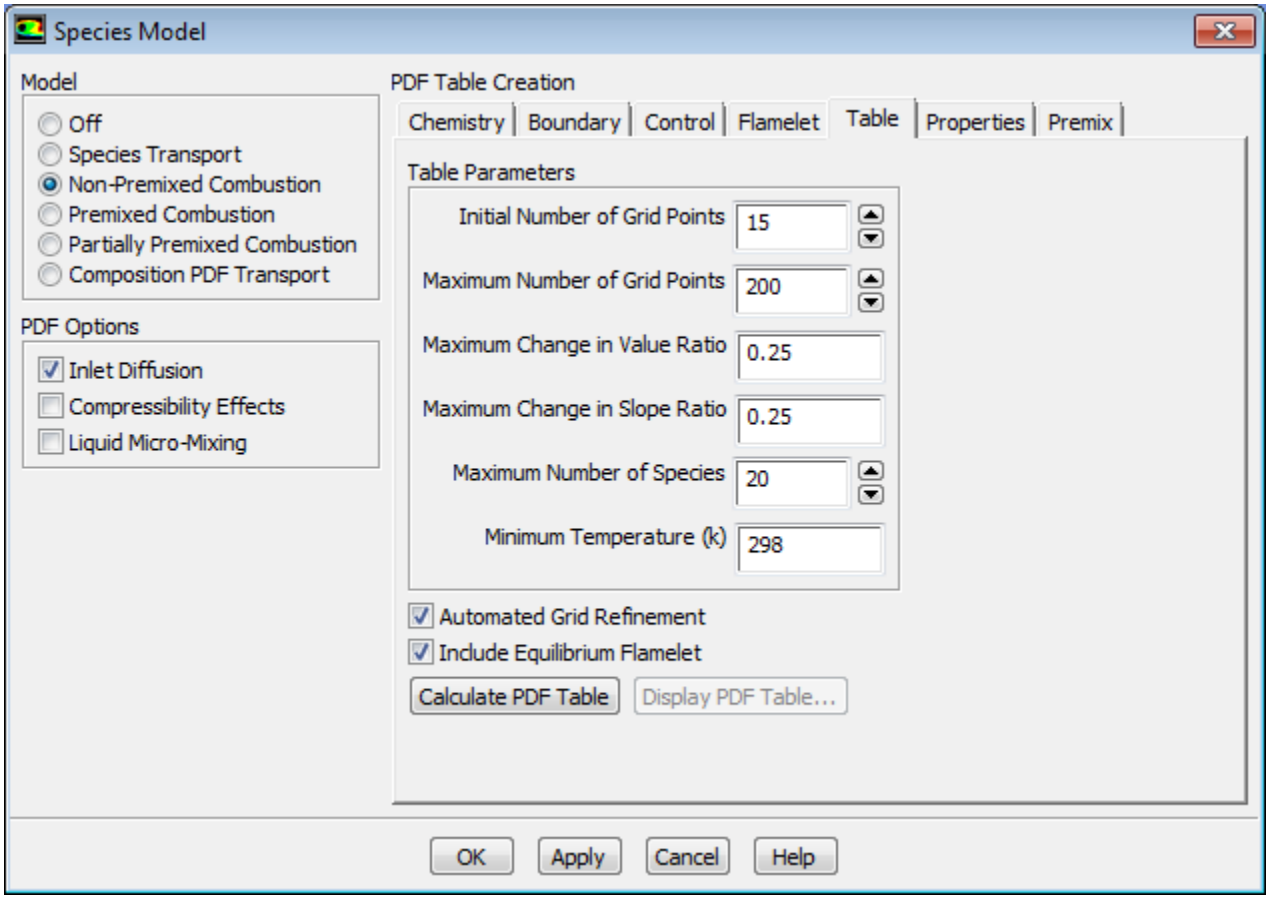
ANSYS Fluent requires additional inputs that are used in the creation of the look-up tables. Several of these inputs control the number of discrete values for which the look-up tables will be computed. These parameters are input in the **Table** tab of the **Species Model** dialog box. When **Automated Grid Refinement** is enabled, you will specify the table parameters displayed in Figure 15.22: The Species Model Dialog Box (Table) Tab Displaying Automated Grid Refinement (p. 1064). If **Automated Grid Refinement** is disabled, you will specify the table parameters displayed in Figure 15.23: The Species Model Dialog Box (Table) Tab Excluding Automated Grid Refinement (p. 1065).

Note

Automated Grid Refinement is not available with two mixture fractions.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.22: The Species Model Dialog Box (Table) Tab Displaying Automated Grid Refinement



The look-up table parameters when **Automated Grid Refinement** is enabled are as follows:

Initial Number of Grid Points
specifies the number of grid points for the resolution of the mean mixture fraction, mixture fraction variance, and mean enthalpy (for non-adiabatic systems).

Maximum Number of Grid Points
specifies the maximum number of grid points used for tabulation. The grid refinement procedure will stop inserting the points when either the change in value and slope between successive points is within tolerance or the maximum number of grid points are generated.

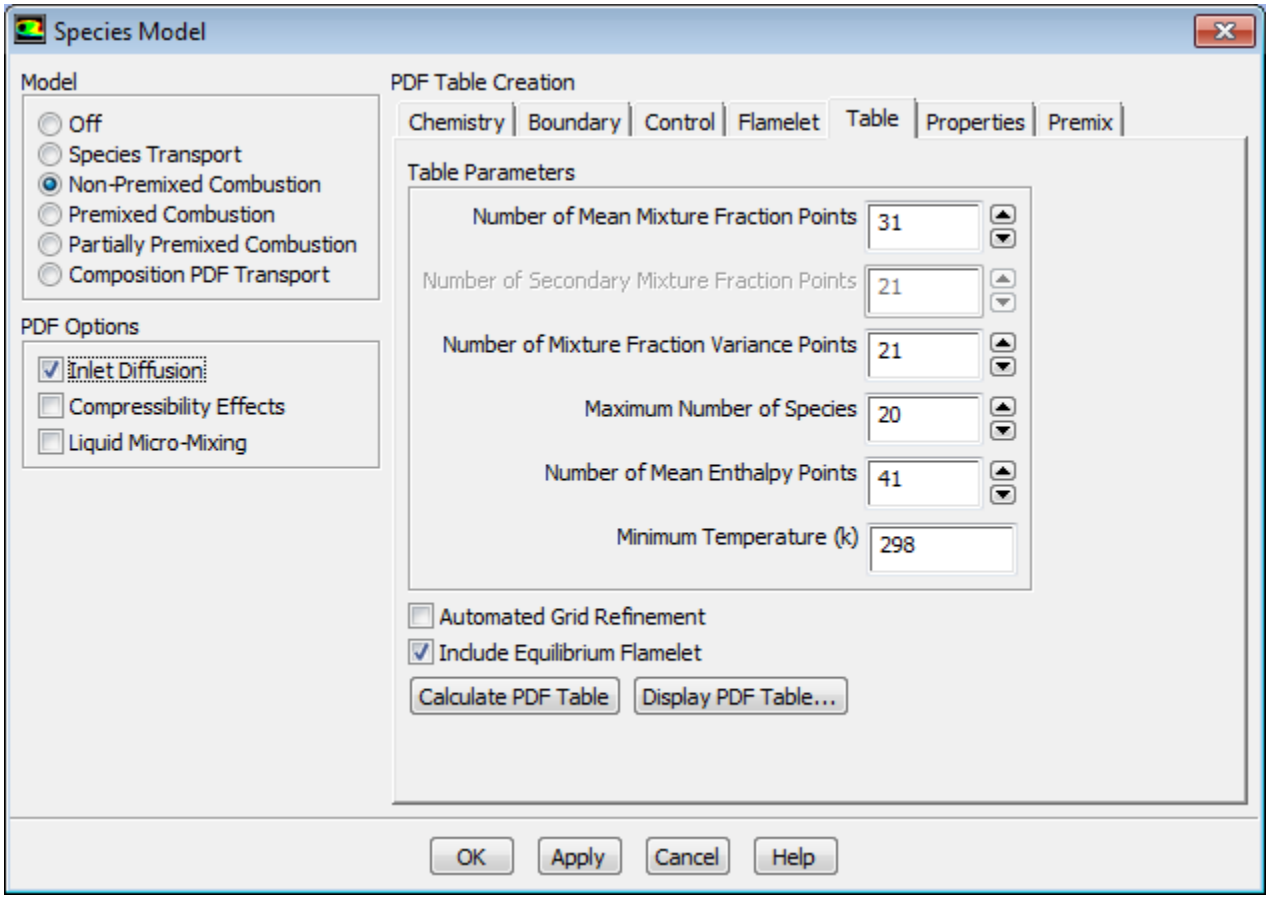
Maximum Change in Value Ratio
specifies the maximum allowable change in value of table variables between successive grid points as specified by Equation 8.27 in the Theory Guide.

Maximum Change in Slope Ratio
specifies the maximum change in the slope of table variables between successive grid points as specified by Equation 8.28 in the Theory Guide.

Maximum Number of Species
is the maximum number of species stored in the lookup tables.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.23: The Species Model Dialog Box (Table) Tab Excluding Automated Grid Refinement



The look-up table parameters when **Automated Grid Refinement** is disabled are as follows:

Number of Mean Mixture Fraction Points

is the number of discrete values of $\overline{f}$ at which the look-up tables will be computed. For a two-mixture-fraction model, this value is the number of points in the instantaneous state profile used to compute the PDF if you choose the β PDF model (see Tuning the PDF Parameters for Two-Mixture-Fraction Calculations (p. 1084)). Increasing the number of points will yield a more accurate PDF shape, but the calculation will take longer. The mean mixture fraction points will be automatically clustered around the stoichiometric mixture fraction value.

Number of Mixture Fraction Variance Points

is the number of discrete values of $\overline{f'^2}$ at which the look-up tables will be computed. Lower resolution is acceptable because the variation along the $\overline{f'^2}$ axis is, in general, slower than the variation along the $\overline{f}$ axis of the look-up tables. This option is available only when no secondary stream has been defined.

Number of Secondary Mixture Fraction Points

is the number of discrete values of p_{sec} at which the look-up tables will be computed. Like the **Number of Mean Mixture Fraction Points**, ANSYS Fluent will use the **Number of Secondary Mixture Fraction Points** to compute the equilibrium state-relation if you choose the β PDF option (see Tuning the PDF Parameters for Two-Mixture-Fraction Calculations (p. 1084)) for a two-mixture-fraction model. A larger number of points will give a more accurate shape for the PDF, but with a longer calculation time. This option is available only when a secondary stream has been defined.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Maximum Number of Species

is the maximum number of species that will be included in the look-up tables. The maximum number of species that can be included is 200. ANSYS Fluent will automatically select the species with the largest mole fractions to include in the PDF table. Note that the PDF table values of density and specific heat are pre-calculated with all the species, and hence the convergence behavior of ANSYS Fluent will not be affected by the input for the **Maximum Number of Species**. Hence, to keep table sizes small, you should set the **Maximum Number of Species** to only include the species that you are interested in postprocessing.

Number of Mean Enthalpy Points

is the number of discrete values of enthalpy at which the look-up tables will be computed. This input is required only if you are modeling a non-adiabatic system. The number of points required will depend on the chemical system that you are considering, with more points required in high heat release systems (for example, hydrogen/oxygen flames). This option is not available with the unsteady flamelet model.

Minimum Temperature

is used to determine the lowest temperature for which the look-up tables are generated. Your input should correspond to the minimum temperature expected in the domain (for example, an inlet or wall temperature). The minimum temperature should be set 10–20 K below the minimum system temperature. This option is available only if you are modeling a non-adiabatic system. This option is not available with the unsteady flamelet model.

Include Equilibrium Flamelet

specifies that an equilibrium flamelet (that is, $\chi=0$) will be generated in ANSYS Fluent and appended to the flamelet library before the PDF table is calculated. This option is available when generating or importing multiple flamelets, as well as when a single flamelet is considered. In the latter case, the PDF table will consist of two scalar dissipation slices, namely the equilibrium slice at $\chi=0$, and the flamelet slice. This option is not available with the equilibrium chemistry model or the unsteady diffusion flamelet model.

When you are satisfied with your inputs, click **Calculate PDF Table** to generate the look-up tables.

The computations performed for a single-mixture-fraction calculation culminate in the discrete integration of Equation 8.16 (or Equation 8.24 in the Theory Guide) as represented in Figure 8.5: Logical Dependence of Averaged Scalars on Mean Mixture Fraction, the Mixture Fraction Variance, and the Chemistry Model (Adiabatic, Single-Mixture-Fraction Systems) (or Figure 8.6: Logical Dependence of Averaged Scalars on Mean Mixture Fraction, the Mixture Fraction Variance, Mean Enthalpy, and the Chemistry Model (Non-Adiabatic, Single-Mixture-Fraction Systems) in the Theory Guide). For a two-mixture-fraction calculation, ANSYS Fluent will calculate the physical properties using Equation 8.14 or its adiabatic equivalent. The computation time will be shortest for adiabatic single-mixture-fraction equilibrium calculations and longest for non-adiabatic calculations involving multiple flamelet generation. Below, sample outputs are shown for an adiabatic single-mixture-fraction equilibrium calculation and a non-adiabatic calculation with flamelets:

```
Generating PDF lookup table
 Type of the PDF Table: Adiabatic Table (Two Streams)
Calculating table .....

1271 points calculated
22 species added
PDF Table successfully generated!

Generating PDF lookup table
 Type of the PDF Table: Nonadiabatic Table with Strained Flamelet Model (Two Streams)
 Calculating table .....
 calculating temperature limits .....
 calculating temperature limits .....
 calculating scalar dissipation slices .....
  - scalar dissipation slice  9
 calculating equilibrium slice .....
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
 Performing PDF integrations.....

 16810 points calculated
 17 species added
 PDF Table successfully generated!
Initializing PDF table arrays and structures.
```

Important

Note that there is a significant difference in run time between the one-mixture fraction model and the two-mixture fraction model. In the one-mixture fraction model, the PDF table contains the mean data of density, temperature, and specific heats, and is three-dimensional for an equilibrium nonadiabatic case (mean mixture fraction, mixture fraction variance, and mean heat loss). For this case, ANSYS Fluent updates properties every flow iteration. In the case of the two-mixture fraction model, only the instantaneous state relationships are stored and mean properties are calculated from these by performing PDF integrations in every cell of the ANSYS Fluent simulation. Since this is computationally expensive, ANSYS Fluent provides the option of only updating properties after a specified number of iterations.

For a two-mixture fraction model, you can also refer to Full Tabulation of the Two-Mixture-Fraction Model (p. 1067) for more information.

After completing the calculation at the specified number of mixture fraction points, ANSYS Fluent reports that the calculation succeeded. In a single-mixture-fraction case, the resulting look-up tables take the form illustrated in Figure 8.8: Visual Representation of a Look-Up Table for the Scalar (Mean Value of Mass Fractions, Density, or Temperature) as a Function of Mean Mixture Fraction and Mixture Fraction Variance in Adiabatic Single-Mixture-Fraction Systems in the Theory Guide (or Figure 8.10: Visual Representation of a Look-Up Table for the Scalar as a Function of Mean Mixture Fraction and Mixture Fraction Variance and Normalized Heat Loss/Gain in Non-Adiabatic Single-Mixture-Fraction Systems, for non-adiabatic systems). These look-up tables can be plotted using the available graphics tools, as described in Postprocessing the Look-Up Table Data (p. 1068).

Note that in non-adiabatic calculations, the console window will report that the temperature limits and enthalpy slices have been calculated.

For a two-mixture-fraction case, the resulting look-up tables take the form illustrated in Figure 8.9: Visual Representation of a Look-Up Table for the Scalar ϕ_I as a Function of Fuel Mixture Fraction and Secondary Partial Fraction in Adiabatic Two-Mixture-Fraction Systems in the Theory Guide (or Figure 8.11: Visual Representation of a Look-Up Table for the Scalar ϕ_I as a Function of Fuel Mixture Fraction and Secondary Partial Fraction, and Normalized Heat Loss/Gain in Non-Adiabatic Two-Mixture-Fraction Systems, for non-adiabatic systems).

15.7.1. Full Tabulation of the Two-Mixture-Fraction Model

The default algorithm for the two-mixture-fraction model is to perform PDF integrations of the equilibrium state relationships at run time. Since these are multi-dimensional integrals, the default two-mixture-fraction model can be computationally demanding.

Alternatively, you may want to pre-compute these integrations and create 4D look-up tables for adiabatic simulations, or 5D tables for non-adiabatic simulations. Such high-dimensional tables are computationally expensive to build, and may require large memory and disk storage, but can offer substantial improvement in run time speed.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The option to create a fully-tabulated two-mixture-fraction table is available in cases with the two-mixture-fraction model enabled, via the text command:

```
define/models/species/full-tabulation?
```

If you are modeling pollutant formation, note that the full tabulation option is not compatible with the **mixture-fraction** option for **PDF Mode** in the **Turbulence Interaction Mode** tab settings. See Setting Turbulence Parameters (p. 1166) for details on Turbulence Interaction options for pollutant modeling.

15.7.2. Stability Issues in Calculating Chemical Equilibrium Look-Up Tables

Complex chemistry and non-adiabatic effects may make the equilibrium calculation more time-consuming and difficult. In some instances the equilibrium calculation may even fail. You may be able to eliminate any difficulties that you encounter by trying the calculation as an adiabatic system. Adiabatic system calculations are generally quite straightforward and can provide valuable insight into the optimal inputs to the non-adiabatic calculation.

Additional stability issues may arise for solid or heavy liquid fuels that have been defined using the empirical fuel approach. You may find that, for rich fuel mixtures, the equilibrium calculation produces very low temperatures and eventually fails. This indicates that strong endothermic reactions are taking place and the mixture is not able to sustain them. In this situation, you may need to raise the heating value of the fuel until ANSYS Fluent produces acceptable results. Provided that your fuel will be treated as a liquid or solid (coal) fuel, you can maintain the desired heating value in your ANSYS Fluent simulation. This is accomplished by defining the difference between the desired and the adjusted heating values as latent heat (in the case of combusting solid fuel) or heat of pyrolysis (in the case of liquid fuel).

15.7.3. Saving the Look-Up Tables

The look-up tables may be stored in a file that you can read back into later sessions of ANSYS Fluent. The look-up tables should be saved before you exit from the current ANSYS Fluent session.

File → **Write** → **PDF...**

By default, the file will be saved as formatted (ASCII, or text). To save a binary (unformatted) file, turn on the **Write Binary Files** option in the **Select File** dialog box.

15.7.4. Postprocessing the Look-Up Table Data

It is important for you to view your temperature and species tables to ensure that they are adequately but not excessively resolved. Inadequate resolution will lead to inaccuracies, and excessive resolution will lead to unnecessarily slow calculation times.

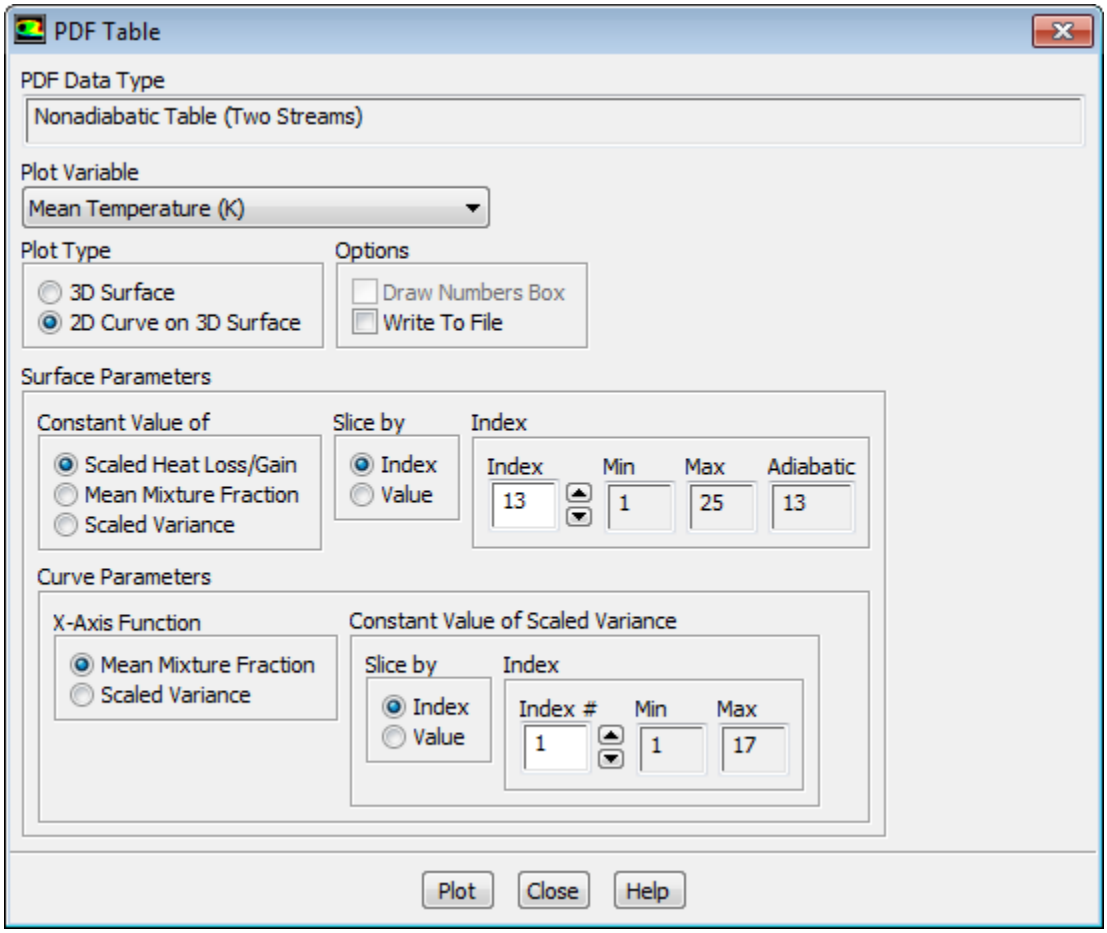
After a PDF table has been generated or read into ANSYS Fluent, you can display 2D plots and 3D surfaces showing the variation of species mole fraction, density, or temperature with the mean mixture fraction, mixture fraction variance, or enthalpy using the **PDF Table** dialog box (for example, Figure 15.24: The PDF Table Dialog Box (Non-Adiabatic Case With Flamelets) (p. 1069)). The **PDF Table** dialog box can be accessed in one of two ways:

- by clicking the **Display PDF Table...** button in the **Table** tab of the **Species Model** dialog box (as shown in Figure 15.23: The Species Model Dialog Box (Table) Tab Excluding Automated Grid Refinement (p. 1065))
- by using the following path:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Postprocessing → Model Specific → PDF Table...

Figure 15.24: The PDF Table Dialog Box (Non-Adiabatic Case With Flamelets)



To display the look-up tables graphically, use the following procedure:

1. In the **PDF Table** dialog box, in the **Plot Variable** drop-down list you can select temperature, density, or species mole fraction as the variable to be plotted.

2. (multiple flamelets only) Specify the value of the **Scalar Dissipation**. In the case of non-adiabatic flamelets, there is the additional parameter of mean enthalpy. In addition to varying the mean enthalpy and mean mixture fraction, you can vary the display of the PDF table by changing the value of the scalar dissipation, which gives the table a fourth "dimension".

3. Specify the **Plot Type** as either **3D Surface** or **2D Curve on 3D Surface**. In the equilibrium model, a 2D curve is a slice of a 3D surface, and therefore some options selected for a 3D surface may impact the display of a 2D curve.

   - For a 3D surface:

     a. Enable or disable **Draw Numbers Box** under **Options**. When this option is turned on, the display will include a wireframe box with the numerical limits in each coordinate direction.

     b. (non-adiabatic cases only) Under **Surface Parameters**, specify the discrete independent variable to be held constant in the look-up table (**Constant Value of**).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– For a single-mixture-fraction case, select **Scaled Heat Loss/Gain** ($\overline{H}$), **Mean Mixture Fraction** ($\overline{f}$), or **Scaled Variance** ($\overline{f'^2}$). For any mean mixture fraction $\overline{f}$, the variance varies between a minimum of 0 and a maximum of $\overline{f}\left(1-\overline{f}\right)$. In order to view the mixture fraction variance, it is normalized by Equation 15.7 (p. 1070) so that for any mean mixture fraction the scaled variance ranges from 0 to 0.25.

$$\overline{f_s'^2}=0.25\frac{\overline{f'^2}}{\overline{f}\left(1-\overline{f}\right)} \tag{15.7}$$

– For a two-mixture-fraction case, the **Scaled Heat Loss/Gain** is the only available option.

c. (non-adiabatic cases only) Specify whether the 3D array of data points available in the look-up table will be sliced by **Index** or **Value** under **Slice by**.

– If you selected **Index**, specify the discretization **Index** of the variable that is being held constant. The range of integer values that you are allowed to choose from is displayed under **Min** and **Max**, and is equivalent to the number of points specified for that variable in the **Table** tab of the **Species Model** dialog box (see Calculating the Look-Up Tables (p. 1063)). If you specified to hold the enthalpy (**Scaled Heat Loss/Gain**) constant, the enthalpy slice index corresponding to the adiabatic case will be displayed in the **Adiabatic** field.

– If you selected **Value**, specify the numerical **Value** of the variable that is being held constant. The range of values that you can specify is displayed under **Min** and **Max**.

- For a 2D curve on a 3D surface:

a. Specify whether you want to write the plot data to a file by toggling **Write To File** under **Options**.

b. Under **Curve Parameters**, specify the **X-Axis Function** against which the plot variable will be displayed.

– For an adiabatic single-mixture-fraction case, select **Mean Mixture Fraction** ($\overline{f}$), or **Scaled Variance** ($\overline{f'^2}$).

– For a non-adiabatic single-mixture-fraction case, the options will depend on what was selected under **Constant Value of** under **Surface Parameters**, but will include two of the following: **Scaled Heat Loss/Gain** ($\overline{H}$), **Mean Mixture Fraction**, and **Scaled Variance**.

– For a two-mixture-fraction case, select **Fuel Mixture Fraction** (f_{fuel}) or **Secondary Partial Fraction** (p_{sec}).

c. Specify the type of discretization (that is, how the look-up table data will be sliced) for the variable that is being held constant (under **Constant Value of Mean Mixture Fraction**, **Constant Value of Scaled Variance**, and so on). Note that for non-adiabatic cases, each 3D surface slice contains a full set of 2D slices.

– If you selected **Index** under **Slice by**, specify the discretization **Index** of the variable that is being held constant. The range of integer values that you are allowed to choose from is displayed under **Min** and **Max**, and is equivalent to the number of points specified for that variable in the **Table** tab of the **Species Model** dialog box (see Calculating the Look-Up Tables (p. 1063)).

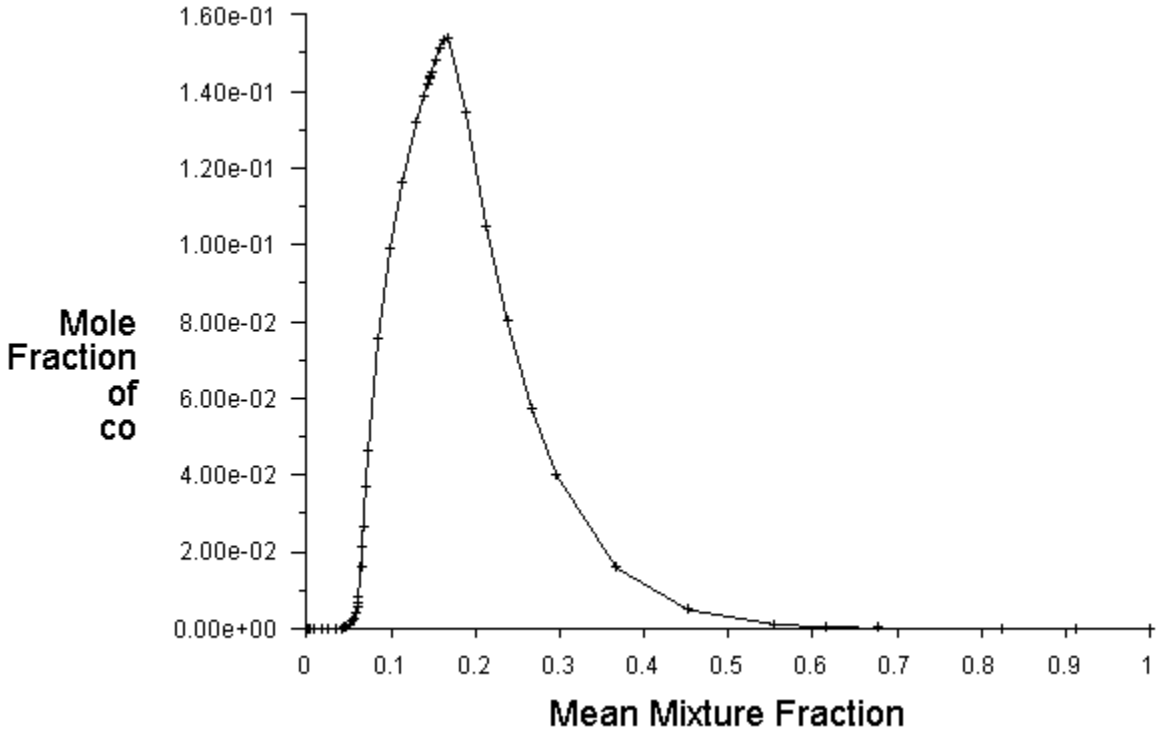
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– If you selected **Value** under **Slice by**, specify the numerical **Value** of the variable that is being held constant. The range of values that you can specify is displayed under **Min** and **Max**.

4. Write or display the look-up table results. If you have turned on the **Write To File** option for a 2D plot, click **Write** and specify a name for the file in The Select File Dialog Box (p. 19). Otherwise, click **Plot** or **Display** as appropriate to display a 2D plot or 3D surface in the graphics window.

Figure 15.25: Mean Species Mole Fraction Derived From an Equilibrium Chemistry Calculation (p. 1071) and Figure 15.26: Mean Temperature Derived From an Equilibrium Chemistry Calculation (p. 1072) show examples of 2D plots derived for a very simple hydrocarbon system.

Figure 15.25: Mean Species Mole Fraction Derived From an Equilibrium Chemistry Calculation



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.26: Mean Temperature Derived From an Equilibrium Chemistry Calculation

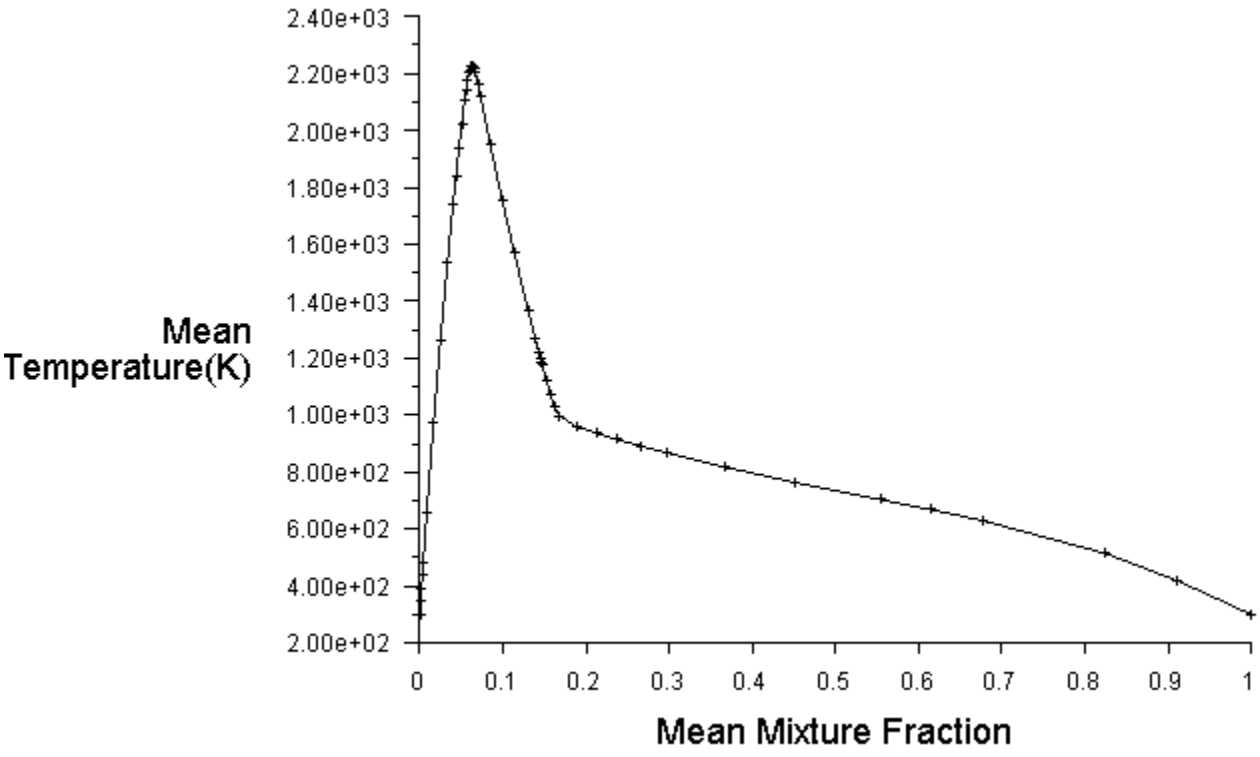
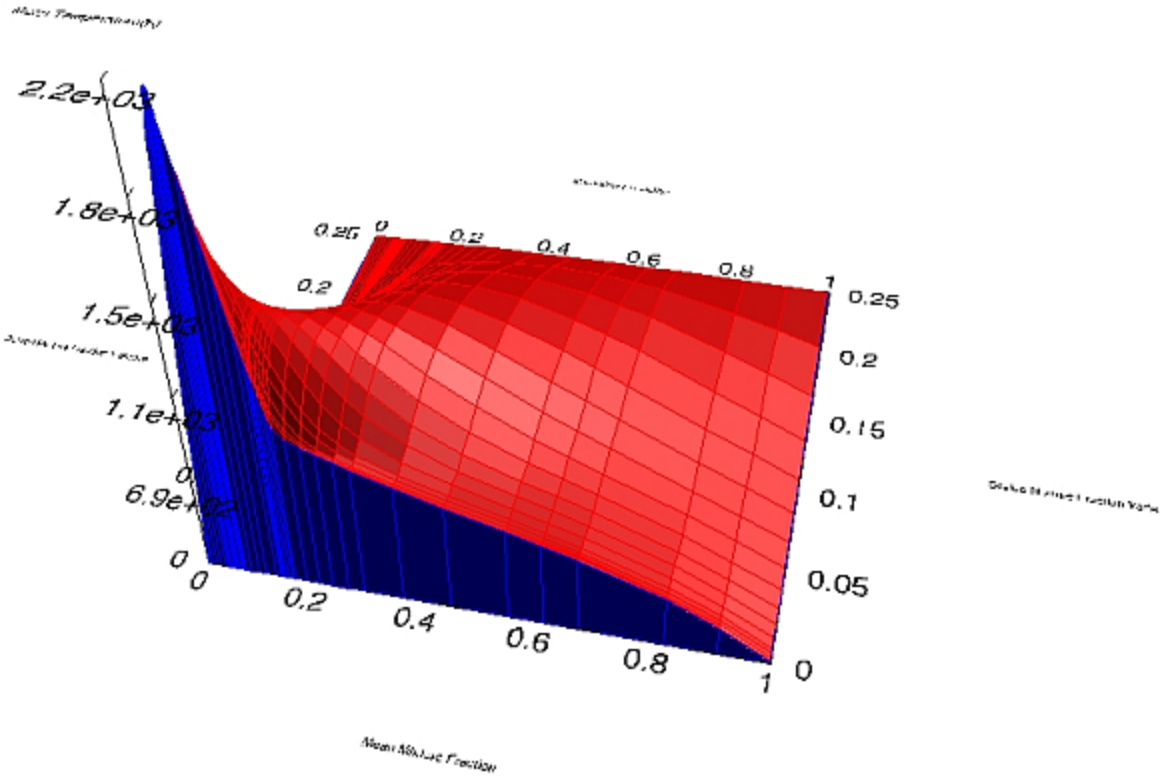


Figure 15.27: 3D Plot of Look-Up Table for Temperature Generated for a Simple Hydrocarbon System (p. 1073) shows an example of a 3D surface derived for the same system.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.27: 3D Plot of Look-Up Table for Temperature Generated for a Simple Hydrocarbon System



15.8. Standard Files for Diffusion Flamelet Modeling

The standard flamelet file format can be used to read and write non-premixed (diffusion) flamelets. The data structure for the standard flamelet file format is based on keywords that precede each data section. If any of the keywords in your flamelet data file do not match the supported keywords, you will have to manually edit the file and change the keywords to one of the supported types. (The ANSYS Fluent flamelet filter is case-insensitive, so you need not worry about capitalization within the keywords.)

The following keywords are supported by the ANSYS Fluent filter:

Description	Keyword
Header section	HEADER
Main body section	BODY
Number of species	NUMOFSPECIES
Number of grid points	GRIDPOINTS
Pressure	PRESSURE
Strain rate for diffusion flamelet generated in physical space	STRAINRATE
Stoichiometric scalar dissipation for diffusion flamelet	STOICH_SCADIS or CHI
Stoichiometric mixture fraction	STOICH_Z
Temperature	TEMPERATURE and TEMP
Mass fraction	MASSFRACTION-

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Keyword
Mixture fraction	Z
Mole fraction	MOLEFRACTION-

Note

The strain rate output is written if the flamelets are generated in physical space, for example, using ANSYS CHEMKIN.

15.8.1. Sample Standard Diffusion Flamelet File

A sample diffusion flamelet file in the standard format is provided below. Note that not all species are listed in this file.

```
HEADER
STOICH_SCADIS   1.000000E-02
NUMOFSPECIES    60
GRIDPOINTS      50
STOICH_Z        6.618652E-02
PRESSURE        1.013250E+05
BODY
Z
 0.000000000E+00 8.273315430E-03 1.240997314E-02 1.447830200E-02 1.551246643E-02
 1.602954865E-02 1.654663086E-02 2.068328857E-02 2.481994629E-02 3.309326172E-02
 4.136657715E-02 4.963989258E-02 5.377655029E-02 5.791320801E-02 6.204986572E-02
 6.618652344E-02 6.912528540E-02 7.206404735E-02 7.500280931E-02 7.794157127E-02
 7.867626176E-02 7.941095225E-02 8.088033323E-02 8.381909519E-02 8.675785714E-02
 8.969661910E-02 9.263538106E-02 9.557414302E-02 1.014516669E-01 1.073291909E-01
 1.132067148E-01 1.190842387E-01 1.249617626E-01 1.367168104E-01 1.425943343E-01
 1.484718583E-01 1.543493822E-01 1.572881441E-01 1.602269061E-01 1.672799348E-01
 1.743329635E-01 1.884390209E-01 2.166511357E-01 2.448632505E-01 2.730753653E-01
 3.407844408E-01 4.084935163E-01 5.709952975E-01 7.659974350E-01 1.000000000E+00
 TEMPERATURE
 3.000000000E+02 6.268546798E+02 7.802882604E+02 8.544333704E+02 8.908936814E+02
 9.089771589E+02 9.269654154E+02 1.067550740E+03 1.203418117E+03 1.462664100E+03
 1.707202552E+03 1.934669997E+03 2.037622483E+03 2.128475386E+03 2.200397351E+03
 2.244572249E+03 2.254177637E+03 2.245072250E+03 2.220638695E+03 2.186735136E+03
 2.177535664E+03 2.168109526E+03 2.148737393E+03 2.108856496E+03 2.068758176E+03
 2.029154219E+03 1.990673114E+03 1.953782923E+03 1.885112633E+03 1.821857119E+03
 1.761797286E+03 1.703540208E+03 1.647197627E+03 1.543457773E+03 1.496624533E+03
 1.452642813E+03 1.410968313E+03 1.390849359E+03 1.371237774E+03 1.326602075E+03
 1.286669965E+03 1.222176660E+03 1.134221753E+03 1.064763074E+03 1.004219101E+03
 8.845385948E+02 7.871510772E+02 6.152847269E+02 4.605312740E+02 3.000000000E+02
 massfraction-h
 0.000000000E+00 8.470246609E-15 1.689876963E-13 7.823505733E-13 1.604836416E-12
 2.259465253E-12 3.137901769E-12 2.297692554E-11 1.299314819E-10 2.009221085E-09
 3.364446760E-08 4.504321512E-07 1.488600091E-06 4.041919731E-06 9.259044043E-06
 1.764854561E-05 2.446982906E-05 2.948355957E-05 3.067005444E-05 2.858096141E-05
 2.763204904E-05 2.660609671E-05 2.447858861E-05 2.024662524E-05 1.571691024E-05
 1.207106883E-05 9.769042791E-06 8.144795604E-06 5.683704776E-06 3.862438419E-06
 2.529575950E-06 1.577642294E-06 9.493198194E-07 3.597224020E-07 2.397461608E-07
 1.672683885E-07 1.109095223E-07 8.352078253E-08 5.847760877E-08 2.024498078E-08
 6.804126487E-09 1.241975019E-09 1.664800458E-10 3.824650210E-11 1.058872219E-11
 1.103611172E-12 2.070333150E-13 5.521048377E-15 6.116361056E-17 0.000000000E+00
 massfraction-o2
 2.330000000E-01 2.037872969E-01 1.891809523E-01 1.818777426E-01 1.782261116E-01
 1.764002861E-01 1.745744526E-01 1.599674402E-01 1.453586904E-01 1.161277458E-01
 8.685446531E-02 5.764270922E-02 4.337649028E-02 2.986903537E-02 1.792882492E-02
 8.656537748E-03 4.260825775E-03 1.718417261E-03 5.787962652E-04 1.809733781E-04
 1.319197735E-04 9.626119601E-05 5.195145104E-05 1.624530279E-05 5.000207346E-06
 1.625791857E-06 5.935574928E-07 2.314963476E-07 5.242565474E-08 1.388575920E-08
 4.396692299E-09 1.701489188E-09 7.617025651E-10 3.172983452E-10 2.146818725E-10
 1.500353242E-10 1.052693603E-10 8.680804793E-11 7.032480088E-11 3.785587351E-11
 1.627110024E-11 1.631110236E-12 2.316226342E-14 1.904385354E-16 8.159316969E-19
 2.851864505E-21 7.569901388E-23 1.016058744E-25 1.304158252E-26 0.000000000E+00
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
massfraction-oh
. . . . .
. . . . .
. . . . .
```

15.8.2. Missing Species

ANSYS Fluent will check whether all species in the flamelet data file exist in the thermodynamic properties databases `thermo.db`. If any of the species in the flamelet file do not exist, ANSYS Fluent will issue an error message and halt the flamelet import. If this occurs, you can either add the missing species to the database, or remove the species from the flamelet file.

You should not remove a species from the flamelet data file unless its species concentration is very small (10^{-3} or less) throughout the flamelet profile. If you remove a low-concentration species, you will not have the species concentrations available for viewing in the ANSYS Fluent calculation, but the accuracy of the ANSYS Fluent calculation will otherwise be unaffected.

Important

If you choose to remove any species, be sure to also update the number of species (keyword `NUMOFSPECIES`) in the flamelet data file, to reflect the loss of any species you have removed from the file.

If a species with relatively large concentration is missing from the ANSYS Fluent thermodynamic databases, you will have to add it. Removing a high-concentration species from the flamelet file is not recommended.

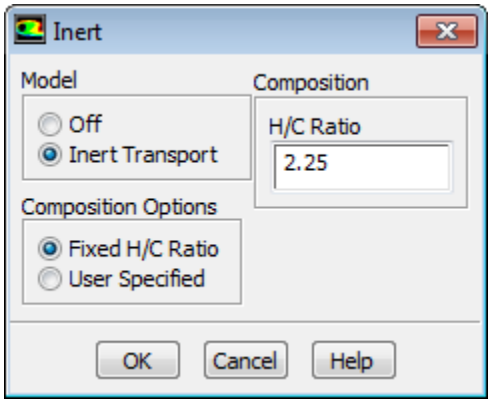
15.9. Setting Up the Inert Model

This section describes how to set up and apply the inert model. For a discussion about the theory, refer to Using the Non-Premixed Model with the Inert Model in the Theory Guide.

To enable the inert model, make sure that the non-premixed or partially premixed model is selected in the **Species Model** dialog box, or that a PDF file is read. Refer to Steps in Using the Non-Premixed Model (p. 1025) and Modeling Partially Premixed Combustion (p. 1099) to learn more about these models.

Setup → Models → Inert Edit...

Figure 15.28: The Inert Model Dialog Box



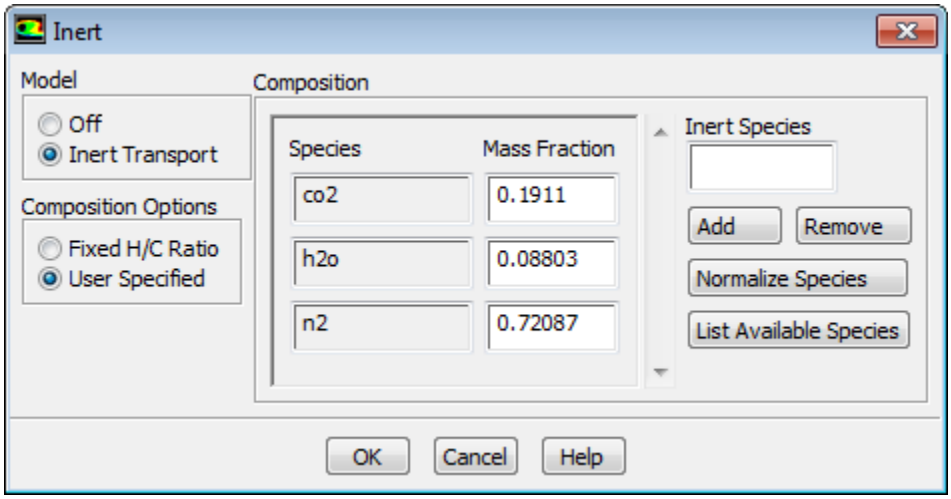
The **Inert** dialog box will be displayed (Figure 15.28: The Inert Model Dialog Box (p. 1075)). To enable this model, select **Inert Transport**. The following steps will walk you through setting up the inert model:

1. Select **Fixed H/C Ratio** as the **Composition Option** if the hydrogen to carbon ratio is known. For example, for methane (CH_4) enter 4 for **H/C Ratio**.

   Setting the H/C ratio assumes that the burned gas resulted from the complete, stoichiometric combustion of that hydrocarbon fuel with air, and the only products of the combustion are CO_2, H_2O and N_2.

2. Select **User Specified** as the **Composition Option** if you want to specify an arbitrary composition for the inert stream, as shown in Figure 15.29: The Inert Model Dialog Box (p. 1076).

   Figure 15.29: The Inert Model Dialog Box



   You can specify your composition stream by adding or removing species if your stream is composed of species other than the default species list.

   a. To add species to the **Species** list, type the chemical formula under **Inert Species** and click **Add**.

   b. Enter the **Mass Fraction** of the newly added species. Continue in this manner until all of the inert species you want to include are shown in the **Species** list.

   c. Make sure the sum of the mass fractions add up to 1. ANSYS Fluent will normalize the species mass fractions for you when you click **Normalize Species**.

   d. To remove a species from the list, type the chemical formula under **Inert Species** and click **Remove**.

   e. To print a list of all species in the thermodynamic database file (`thermo.db`) in the console window, click **List Available Species**.

15.9.1. Setting Boundary Conditions for Inert Transport

You will need to set appropriate boundary conditions at flow inlets and exits for the inert tracer mass fraction, Y_I. The tracer species mass fraction must be between zero and one, with the value of one meaning that all of the material entering the domain comes from the inert stream. The values for flow boundaries are set in the **Inert Stream** field of the inlet boundary condition dialog boxes, under the **Species** tab.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.9.2. Initializing the Inert Stream

The main assumption of the inert model is that the composition of the inert stream does not change with combustion. For some dilutants, this is a very reasonable assumption, however, it is not valid for rich combustion where there is fuel in the exhaust stream. For cases where there is fuel or oxidizer left in the exhaust gas, the accuracy of your results will depend upon taking the fuel or oxidizer species into account when setting initial conditions.

15.9.2.1. Inert Fraction

Initialization of the inert mass fraction Y_I is done in the same way as other variables: by entering in the appropriate value in the **Solution Initialization** task page. Another option for initialization is to patch the value of Y_I in a region of the domain. When the value of Y_I is patched in this way, ANSYS Fluent automatically recalculates the enthalpy field for the current temperature field in order to account for the change in composition.

See Patching Values in Selected Cells (p. 1593) for details about patching values of solution variables.

15.9.2.2. Inert Composition

For combustion calculations that burn a hydrocarbon fuel, ANSYS Fluent provides a straightforward way of setting the initial composition of the inert stream. The inert composition can be set by assuming a ratio of hydrogen to carbon in the following overall oxidation reaction (from Heywood [42] (p. 2889)):

$$C_aH_b+\left(a+\frac{b}{4}\right)\left(O_2+3.773N_2\right)=aCO_2+\frac{b}{2}H_2O+3.773\left(a+\frac{b}{4}\right)N_2 \tag{15.8}$$

which can be rewritten in terms of the ratio of hydrogen to carbon atoms ($y=b/a$) in the fuel as

$$CH_y+\left(1+\frac{y}{4}\right)\left(O_2+3.773N_2\right)=CO_2+\frac{y}{2}H_2O+3.773\left(1+\frac{y}{4}\right)N_2 \tag{15.9}$$

If Equation 15.9 (p. 1077) is solved for the mass fractions of CO_2, H_2O and N_2, the following relations are obtained:

$$\begin{aligned} Y_{CO_2} &= \frac{44.01}{y_{tot}} \\ Y_{H_2O} &= \frac{9.01y}{y_{tot}} \\ Y_{N_2} &= \frac{26.56192y+106.24768}{y_{tot}} \end{aligned} \tag{15.10}$$

where

$$y_{tot}=35.57192y+150.25768$$

Setting the H/C ratio assumes that the burned gas resulted from the complete, stoichiometric combustion of that hydrocarbon fuel, and the only products of the combustion are CO_2, H_2O and N_2. An arbitrary composition for the inert stream can also be specified in the interface.

15.9.3. Resetting Inert EGR

ANSYS Fluent allows burned gases to be converted to an inert gas. This has been designed with in-cylinder combustion in mind to aid the simulation of multiple cycles of such engines using the partially-premixed combustion model with exhaust gas recirculation (EGR). At the end of a combustion stroke, the premixed progress variable of the burnt gases is unity. When fresh charge, with progress variable

of zero, is mixed with the burnt trapped gases, combustion will be initiated unless these trapped gases are converted to inert.

This facility is accessed via the **Dynamic Mesh Events** dialog box, as described in Resetting Inert EGR (p. 722). There, you will need to set the crank angle at which the event occurs (usually shortly before the inlet valves open) and at the fluid zones (usually the combustion chamber) where the burnt gases are converted to inert.

When ANSYS Fluent performs an inert reset, the inert composition is calculated as the stoichiometric composition. For lean mixtures, the stoichiometric inert is mixed with oxidizer, and for rich mixtures the stoichiometric inert is mixed with fuel, so that overall stoichiometry is conserved. This entails:

- Calculating the stoichiometric mixture fraction, which is defined as the mixture where both fuel and oxidizer are completely consumed.
- Calculating the inert species mass fractions as the stoichiometric species mass fractions. Species with mole fractions less than 0.01 are discarded.
- Setting the inert mass fraction and the mixture fraction as follows:
  - Lean mixtures

$$Y_i = Y_{i,old} + \left(1.0 - Y_{i,old}\right) \cdot \tilde{c} \cdot \frac{\tilde{f}}{\tilde{f}_{sto}}$$

$$\tilde{f} = \tilde{f}_{old} \cdot \left(1.0 - \tilde{c}\right)$$

  - Rich mixtures

$$Y_i = Y_{i,old} + \left(1.0 - Y_{i,old}\right) \cdot \tilde{c} \cdot \frac{1.0 - \tilde{f}_{old}}{1.0 - \tilde{f}_{sto}}$$

$$\tilde{f} = \tilde{f}_{old} \cdot \left(1.0 - \tilde{c}\right) + \tilde{c} \cdot \frac{\tilde{f}_{old} - \tilde{f}_{sto}}{1.0 - \tilde{f}_{sto}}$$

    where Y_i is the inert mass fraction, f is the mixture fraction, c is the reaction progress, subscript *sto* denotes stoichiometric, subscript *old* denotes values before inert reset, tilde superscript denotes Favre averaging.

- Setting the reaction progress to zero.
  - For the ECFM model, flame area density is set to zero.
  - For the G-Equation model, G (the mean distance from the flame front) is set to a negative value.
- Since there will be some small errors in the species mass fractions from before the EGR reset, the temperature will change due to the change in the specific heat. By default, ANSYS Fluent adjusts the enthalpy so that temperature is unchanged after EGR reset. This may be limited by the PDF table temperature limits for the new mixture fraction, in which case the temperature after inert reset will not equal the temperature before the inert reset event.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.10. Defining Non-Premixed Boundary Conditions

For information about defining non-premixed boundary conditions, see the following sections:

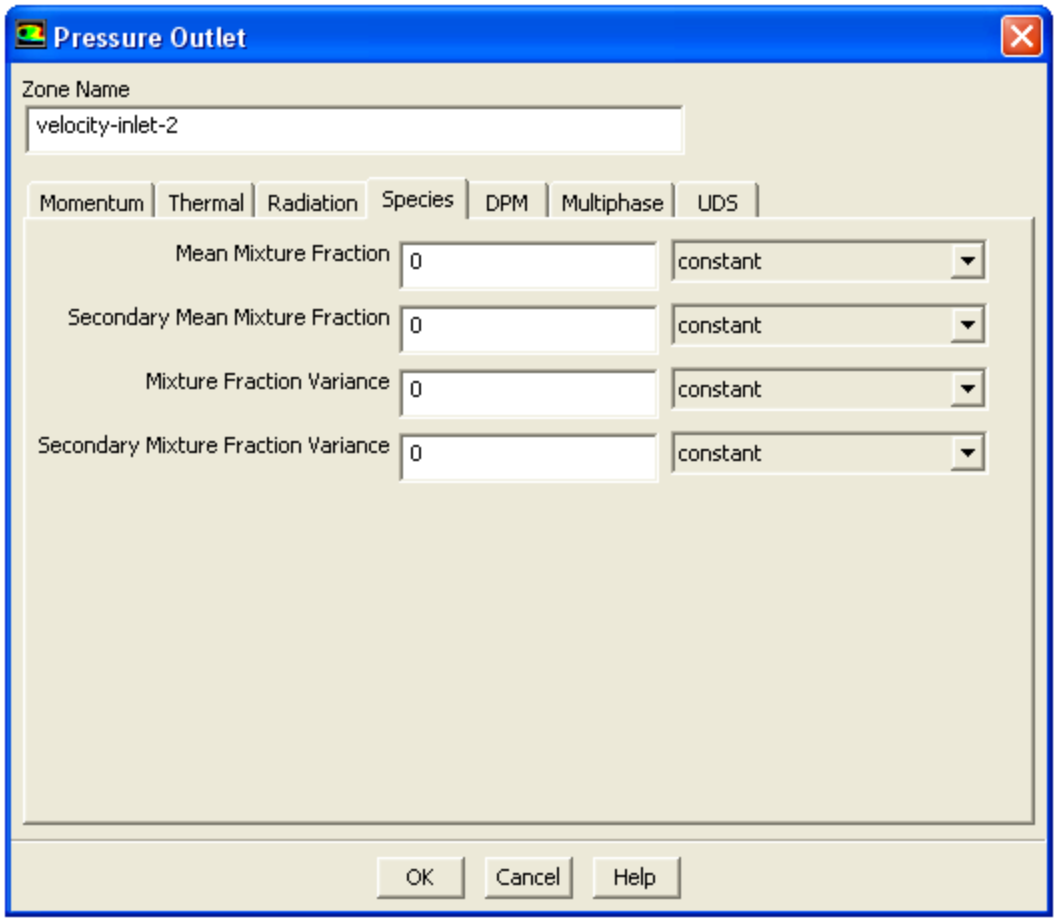
15.10.1. Input of Mixture Fraction Boundary Conditions
15.10.2. Diffusion at Inlets
15.10.3. Input of Thermal Boundary Conditions and Fuel Inlet Velocities

15.10.1. Input of Mixture Fraction Boundary Conditions

When the non-premixed combustion model is used, flow boundary conditions at inlets and exits (that is, velocity or pressure, turbulence intensity) are defined in the usual way. Species mass fractions at inlets are not required. Instead, you define values for the mean mixture fraction, $\overline{f}$, and the mixture fraction variance, $\overline{f'^2}$, at inlet boundaries. (For problems that include a secondary stream, you will define boundary conditions for the mean secondary partial fraction and its variance as well as the mean fuel mixture fraction and its variance.) These inputs provide boundary conditions for the conservation equations you will solve for these quantities. The inlet values are supplied in the boundary conditions task page, under the available tabs, for the selected inlet boundary (for example, Figure 15.30: The Velocity Inlet Dialog Box Showing Mixture Fraction Boundary Conditions (p. 1079)).

Setup → Boundary Conditions

Figure 15.30: The Velocity Inlet Dialog Box Showing Mixture Fraction Boundary Conditions



Click the **Species** tab and specify the **Mean Mixture Fraction** and **Mixture Fraction Variance** (and the **Secondary Mean Mixture Fraction** and **Secondary Mixture Fraction Variance**, if you are using two mixture fractions). In general, the inlet value of the mean fractions will be 1.0 or 0.0 at flow inlets: the mean fuel mixture fraction will be 1.0 at fuel stream inlets and 0.0 at oxidizer or secondary stream inlets; the mean secondary mixture fraction will be 1.0 at secondary stream inlets and 0.0 at fuel or oxidizer inlets. The fuel or secondary mixture fraction will lie between 0.0 and 1.0 only if you are modeling flue gas recycle, as illustrated in Figure 8.15: Using the Non-Premixed Model with Flue Gas Recycle and discussed in Definition of the Mixture Fraction in the Theory Guide. The fuel or secondary mixture fraction variance can usually be taken as zero at inlet boundaries.

15.10.2. Diffusion at Inlets

In some cases, you may want to include the diffusive transport of mixture fraction through the inlets of your domain. You can do this by enabling inlet mixture-fraction diffusion. By default, ANSYS Fluent excludes the diffusion flux of mixture fraction at inlets. To enable inlet diffusion, use either the

```
define/models/ species/inlet-diffusion?
```

text command, or the **Species Model** dialog box (Figure 15.1: Defining Equilibrium Chemistry (p. 1027))

Setup → **Models** → **Species** **Edit...**

and enable the **Inlet Diffusion** option.

15.10.3. Input of Thermal Boundary Conditions and Fuel Inlet Velocities

If your model is non-adiabatic, you should specify the **Temperature** at the flow inlets. Recall that the inlet temperatures were requested during the table construction in the **Chemistry** tab of the **Species Model** dialog box, and were used in the construction of the look-up tables. The inlet temperatures for each fuel, oxidizer, and secondary inlet in your non-adiabatic model should be defined, in addition, as boundary conditions in ANSYS Fluent. It is acceptable for the inlet temperature boundary conditions to differ slightly from those you input for the look-up table calculations. If the inlet temperatures differ significantly from those in the **Chemistry** tab, however, your look-up tables may provide inaccurate interpolation. This is because the discrete points in the look-up tables were clustered around a finite range of the temperatures defined in the **Chemistry** tab, and your input in the ANSYS Fluent inlet boundary condition may fall outside this range.

Wall thermal boundary conditions should also be defined for non-adiabatic non-premixed combustion calculations. You can use any of the standard conditions available in ANSYS Fluent, including specified wall temperature, heat flux, external heat transfer coefficient, or external radiation. If radiation is to be included within the domain, the wall emissivity should be defined as well. See Thermal Boundary Conditions at Walls (p. 370) for details about thermal boundary conditions at walls.

15.11. Defining Non-Premixed Physical Properties

When you use the non-premixed combustion model, the material used for all fluid zones is automatically set to **pdf-mixture**. This material is a special case of the mixture material concept discussed in Mixture Materials (p. 959). The constituent species of this mixture are the species that were calculated in the PDF look-up table creation; you cannot change them directly. When the non-premixed model is used, heat capacities, molecular weights, and enthalpies of formation for each species considered are extracted from the chemical database, so you will not modify any properties for the constituent species in the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PDF mixture. For the PDF mixture itself, the mean density and mean specific heat are determined from the look-up tables.

The physical property inputs for a non-premixed combustion problem are therefore only the transport properties (viscosity, thermal conductivity, and so on) for the PDF mixture. To set these in the Create/Edit Materials Dialog Box (p. 2302), choose **mixture** as the **Material Type**, **pdf-mixture** (the default, and only choice) in the **Mixture Materials** list, and set the desired values for the transport properties.

Setup → **Materials**

See Physical Properties (p. 459) for details about setting physical properties. The transport properties in a non-premixed combustion problem can be defined as functions of temperature, if desired, but not as functions of composition. In practice, since turbulence effects will dominate, it will be of little benefit to include even the temperature dependence of the laminar transport properties.

If you are modeling radiation heat transfer, you will also enter radiation properties, as described in Radiation Properties (p. 515). Composition-dependent absorption coefficients (using the WSGGM) are allowed.

The non-premixed model can also be used with real gas models in ANSYS Fluent, if the **Compressibility Effects** option is enabled in the **Species Model** dialog box. In this case, the density method is based on one of the four real gas models, discussed in Equation of State (p. 537) and Using the Cubic Equation of State Real Gas Models (p. 542).

15.12. Solution Strategies for Non-Premixed Modeling

The non-premixed model setup and solution procedure in ANSYS Fluent differs slightly for single- and two-mixture-fraction problems. Below, an overview of each approach is provided. Note that your ANSYS Fluent case file must always meet the restrictions listed for the non-premixed modeling approach in Restrictions on the Mixture Fraction Approach in the Theory Guide. In this section, details are provided regarding the problem definition and calculation procedures you follow in ANSYS Fluent.

For additional information, see the following sections:

15.12.1. Single-Mixture-Fraction Approach
15.12.2. Two-Mixture-Fraction Approach
15.12.3. Starting a Non-Premixed Calculation From a Previous Case File
15.12.4. Solving the Flow Problem

15.12.1. Single-Mixture-Fraction Approach

For a single-mixture-fraction system, when you have completed the calculation of the PDF look-up tables, you are ready to begin your reacting flow simulation. In ANSYS Fluent, you will solve the flow field and predict the spatial distribution of $\overline{f}$ and $\overline{f'^2}$ (and $\overline{H}$ if the system is non-adiabatic or $\overline{\chi}$ if the system is based on laminar flamelets). ANSYS Fluent will obtain the corresponding values of temperature and individual chemical species mass fractions from the look-up tables.

15.12.2. Two-Mixture-Fraction Approach

When a secondary stream is included, ANSYS Fluent will solve transport equations for the mean secondary partial fraction (p_{sec}) and its variance in addition to the mean fuel mixture fraction and its variance. ANSYS Fluent will then look up the instantaneous values for temperature, density, and individual chemical species in the look-up tables, compute the PDFs for the fuel and secondary streams, and calculate the mean values for temperature, density, and species.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that in order to avoid both inaccuracies and unnecessarily slow calculation times, it is important for you to view your temperature and species tables in ANSYS Fluent to ensure that they are adequately but not excessively resolved.

15.12.3. Starting a Non-Premixed Calculation From a Previous Case File

You can read a previously defined ANSYS Fluent case file as a starting point for your non-premixed combustion modeling. If this case file contains inputs that are incompatible with the current non-premixed combustion model, ANSYS Fluent will alert you when the non-premixed model is turned on and it will turn off those incompatible models. For example, if the case file includes species that differ from those included in the PDF file created by ANSYS Fluent, these species will be disabled. If the case file contains property descriptions that conflict with the property data in the chemical database, these property inputs will be ignored.

Important

PDF files created by prePDF 2 or older are not supported by this version of ANSYS Fluent. The files generated by PrePDF version 3 or newer, are fully compatible.

In the **Species Model** dialog box, select **Non-Premixed Combustion** under the **Model** heading. When you click **OK** in the **Species Model** dialog box, The Select File Dialog Box (p. 19) will immediately appear, prompting you for the name of the PDF file containing the look-up tables created in a previous ANSYS Fluent session. (The PDF file is the file you saved using the **File/Write/PDF...** ribbon tab item after computing the look-up tables.) ANSYS Fluent will indicate that it has successfully read the specified PDF file:

```
Reading "/home/mydirectory/adiabatic.pdf"...
read 5 species (binary c, adiabatic fluent)
pdf file successfully read.
Done.
```

After you read in the PDF file, ANSYS Fluent will inform you that some material properties have changed. You can accept this information; you will be updating properties later on.

You can read in an altered PDF file at any time by using the **File/Read/PDF...** ribbon tab item.

Important

Recall that the non-premixed combustion model is available only when you used the pressure-based solver; it cannot be used with the density-based solvers. Also, the non-premixed combustion model is available only when turbulence modeling is active.

If you are modeling a non-adiabatic system and you want to include the effects of compressibility, re-open the **Species Model** dialog box and turn on **Compressibility Effects** under **PDF Options**. This option tells ANSYS Fluent to update the density according to Equation 15.1 (p. 1033). When the non-premixed combustion model is active, you can enable compressibility effects only in the **Species Model** dialog box. For other models, you will specify compressible flow (**ideal-gas**, **boussinesq**, and so on) in the **Create/Edit Materials** dialog box. See Specifying the Operating Pressure for the System (p. 1033) and Tuning the PDF Parameters for Two-Mixture-Fraction Calculations (p. 1084) for more information about compressibility effects.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.12.3.1. Retrieving the PDF File During Case File Reads

The PDF filename is specified to ANSYS Fluent only once. Thereafter, the filename is stored in your ANSYS Fluent case file and the PDF file will be automatically read into ANSYS Fluent whenever the case file is read. ANSYS Fluent will remind you that it is reading the PDF file after it finishes reading the rest of the case file by reporting its progress in the text (console) window.

Note that the PDF filename stored in your case file may not contain the full name of the directory in which the PDF file exists. The full directory name will be stored in the case file only if you initially read the PDF file through the GUI (or if you typed in the directory name along with the filename when using the text interface). In the event that the full directory name is absent, the automatic reading of the PDF file may fail (since ANSYS Fluent does not know which directory to look in for the file), and you will need to manually specify the PDF file. The safest approaches are to use the GUI when you first read the PDF file or to supply the full directory name when using the text interface.

15.12.4. Solving the Flow Problem

The next step in the non-premixed combustion modeling process in ANSYS Fluent is the solution of the mixture fraction and flow equations. First, initialize the flow. By default, the mixture fraction and its variance have initial values of zero, which is the recommended value; you should generally not set nonzero initial values for these variables. See Initializing the Solution (p. 1591) for details about solution initialization.

Solution → **Initialization**

Next, begin calculations in the usual manner.

Solution → **Run Calculation**

During the calculation process, ANSYS Fluent reports residuals for the mixture fraction and its variance in the `fmean` and `fvar` columns of the residual report:

```
iter       cont       x-vel       y-vel          k     epsilon       fmean       fvar
  28    1.57e-3     4.92e-4     4.80e-4    2.68e-2     2.59e-3     9.09e-1    1.17e+0
  29    1.42e-3     4.43e-4     4.23e-4    2.48e-2     2.30e-3     8.89e-1    1.15e+0
  30    1.28e-3     3.98e-4     3.75e-4    2.29e-2     2.04e-3     8.88e-1    1.14e+0
```

(For two-mixture-fraction calculations, columns for `psec` and `pvar` will also appear.)

15.12.4.1. Under-Relaxation Factors for PDF Equations

The transport equations for the mean mixture fraction and mixture fraction variance are quite stable and high, under-relaxation can be used when solving them. By default, an under-relaxation factor of 1 is used for the mean mixture fraction (and secondary partial fraction) and 0.9 for the mixture fraction variance (and secondary partial fraction variance). If the residuals for these equations are increasing, you should consider decreasing these under-relaxation factors, as discussed in Setting Under-Relaxation Factors (p. 1564).

15.12.4.2. Density Under-Relaxation

One of the main reasons a combustion calculation can have difficulty converging is that large changes in temperature cause large changes in density, which can, in turn, cause instabilities in the flow solution. ANSYS Fluent allows you to under-relax the change in density to alleviate this difficulty. The default

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

value for density under-relaxation is 1, but if you encounter convergence trouble you may want to reduce this to a value between 0.5 and 1 (in the **Solution Controls** task page).

Note that if you are using a real-gas model the solution might converge at a slower rate than when running ideal-gas flow. To improve the convergence performance of your analysis, you need to reduce the underrelaxation for density to values as low as 0.1.

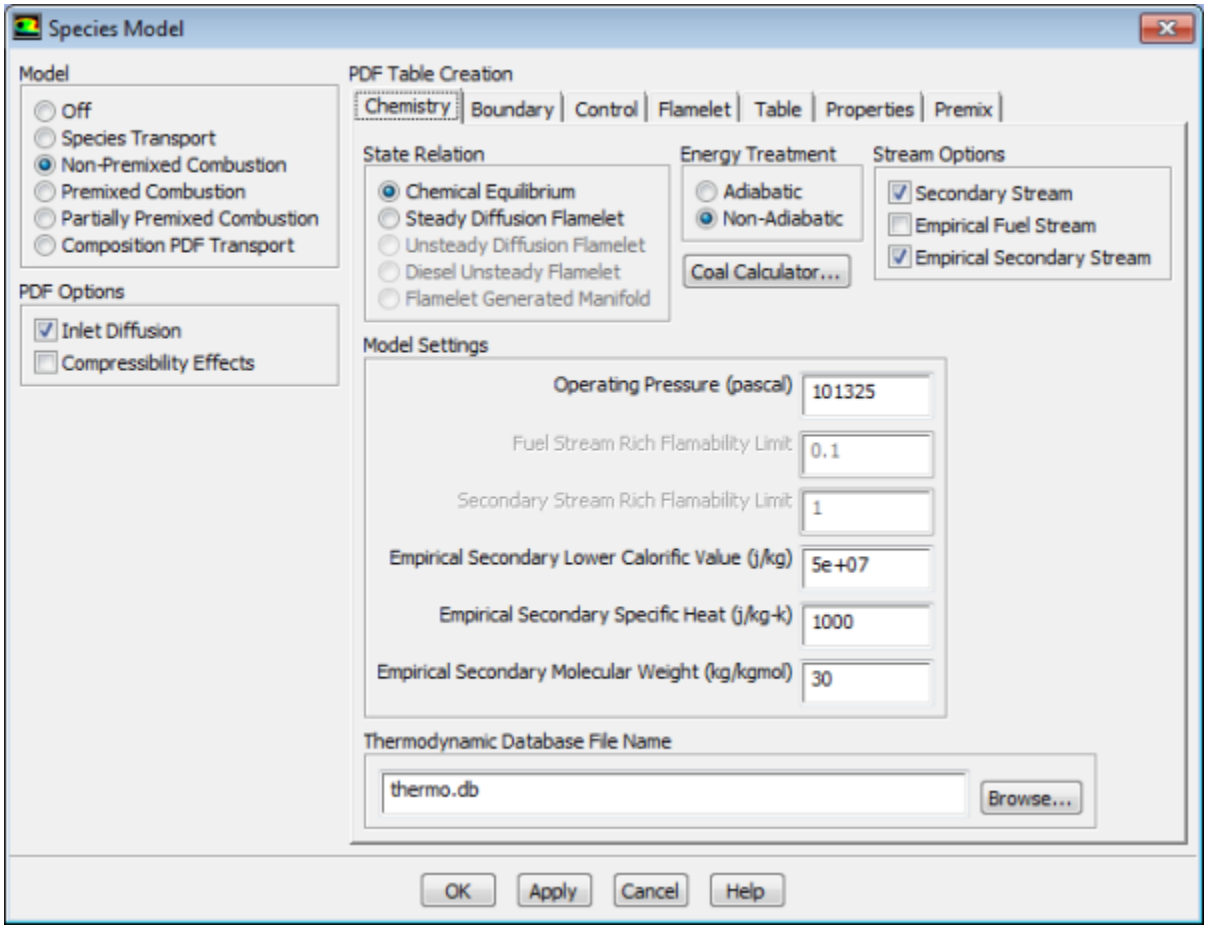
15.12.4.3. Tuning the PDF Parameters for Two-Mixture-Fraction Calculations

For cases that include a secondary stream, the PDF integrations are performed inside ANSYS Fluent.

The parameters for these integrations are defined in the Species Model Dialog Box (p. 2216) (Figure 15.31: The Species Model Dialog Box for a Two-Mixture-Fraction Calculation (p. 1084)).

Setup → Models → Species **Edit...**

Figure 15.31: The Species Model Dialog Box for a Two-Mixture-Fraction Calculation



The parameters are as follows:

Compressibility Effects

(non-adiabatic systems only) tells ANSYS Fluent to update the density, temperature, species mass fraction, and enthalpy from the PDF tables to account for the varying pressure of the system.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.13. Postprocessing the Non-Premixed Model Results

The final step in the non-premixed combustion modeling process is the postprocessing of species concentrations and temperature data from the mixture fraction and flow-field solution data. The following variables are of particular interest:

- **Mean Mixture Fraction** (in the **Pdf...** category)
- **Secondary Mean Mixture Fraction** (in the **Pdf...** category)
- **Mixture Fraction Variance** (in the **Pdf...** category)
- **Secondary Mixture Fraction Variance** (in the **Pdf...** category)
- **Fvar Prod** (in the **Pdf...** category, which is the production term in the mixture fraction variance transport equation)
- **Fvar2 Prod** (in the **Pdf...** category)
- **Scalar Dissipation** (in the **Pdf...** category)
- **PDF Table Adiabatic Enthalpy** (in the **Pdf...** category)
- **PDF Table Heat Loss/Gain** (in the **Pdf...** category)
- **Mass fraction of (species-n)** (in the **Species...** category)
- **Mole fraction of (species-n)** (in the **Species...** category)
- **Molar Concentration of (species-n)** (in the **Species...** category)
- **RMS (species-n) Mass Fraction** (in the **Species...** category)
- **Static Temperature** (in the **Temperature...** category)
- **RMS Temperature** (in the **Temperature...** category)
- **Enthalpy** (in the **Temperature...** category)
- **Probability** (in the **Unsteady Flamelet...** category)
- **Mean Temperature** (in the **Unsteady Flamelet...** category)
- **Mean Mass Fraction of (species-n)** (in the **Unsteady Flamelet...** category)

Important

For the unsteady diffusion flamelet model, mean species mass fractions are displayed for the first fifty species in the flamelet kinetic mechanism.

These quantities can be selected for display in the indicated category of the variable-selection drop-down list that appears in postprocessing dialog boxes. See Field Function Definitions (p. 1931) for their definitions.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In all cases, the species concentrations are derived from the mixture fraction/variance field using the look-up tables. Note that temperature and enthalpy can be postprocessed even when your ANSYS Fluent model is an adiabatic non-premixed combustion simulation in which you have not solved the energy equation. In both the adiabatic and non-adiabatic cases, the temperature is derived from the look-up table.

Figure 15.32: Predicted Contours of Mixture Fraction in a Methane Diffusion Flame (p. 1086) and Figure 15.33: Predicted Contours of CO2 Mass Fraction Using the Non-Premixed Combustion Model (p. 1086) illustrate typical results for a methane diffusion flame modeled using the non-premixed approach.

Figure 15.32: Predicted Contours of Mixture Fraction in a Methane Diffusion Flame

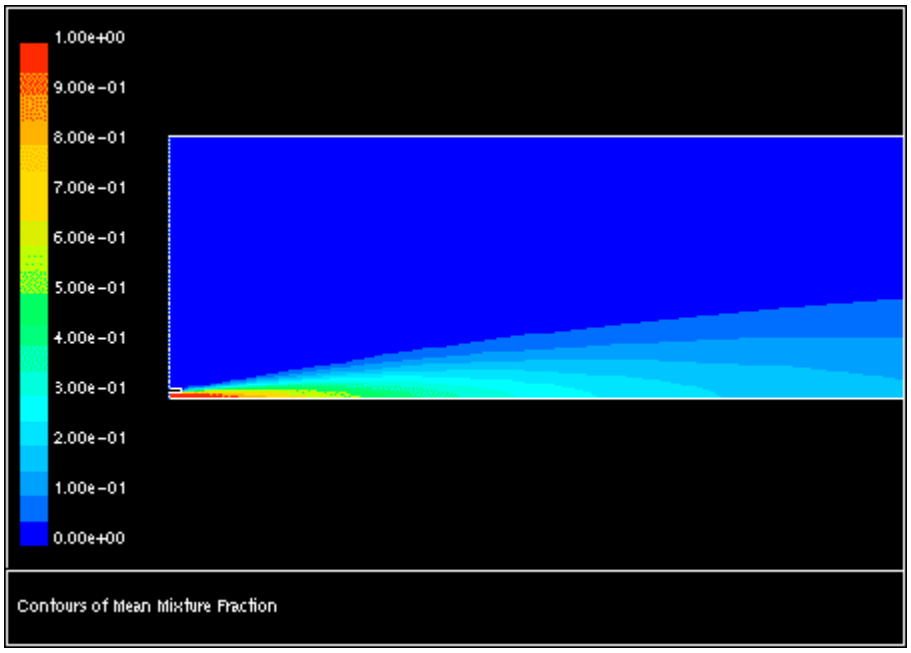
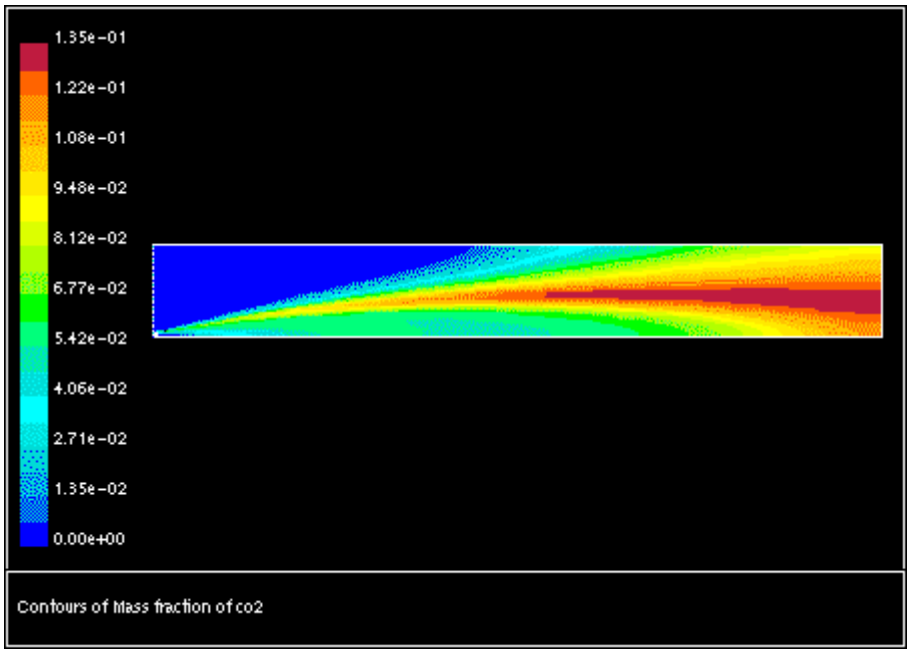


Figure 15.33: Predicted Contours of CO2 Mass Fraction Using the Non-Premixed Combustion Model



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For additional information, see the following section:
15.13.1. Postprocessing for Inert Calculations

15.13.1. Postprocessing for Inert Calculations

ANSYS Fluent provides several additional reporting options for post-processing calculations with the inert model. You can generate graphical plots or alphanumeric reports of the same items that are available with the non-premixed or partially premixed models, and in addition the following variables are available:

- **Inert Mass Fraction**
- **Inert Specific Heat**
- **Inert Density**
- **Inert Enthalpy**
- **Pdf Enthalpy**
- **Pdf Fmean**
- **Pdf Fvar**
- **Mass Fraction of Inert** Y_i

where the mass fraction of the i^{th} inert species, Y_i, is calculated as

$$Y_i = Y_I Y_i^I$$

where Y_I is the inert tracer and Y_i^I is the mass fraction of the i^{th} inert species defined in the **Inert** dialog box.

Note these variables appear in the **Inert...** category.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 16: Modeling Premixed Combustion

ANSYS Fluent has several models to simulate premixed turbulent combustion. For theoretical background on this model, see Premixed Combustion in the Theory Guide. Information about using this model is provided in the following sections:

16.1. Overview and Limitations
16.2. Using the Premixed Combustion Model
16.3. Setting Up the C-Equation and G-Equation Models
16.4. Setting Up the Extended Coherent Flame Model
16.5. Postprocessing for Premixed Combustion Calculations

16.1. Overview and Limitations

In premixed combustion, fuel and oxidizer are mixed at the molecular level prior to ignition. Combustion occurs as a flame front propagating into the unburnt reactants. Examples of premixed combustion include aspirated internal combustion engines, lean-premixed gas turbine combustors, and gas-leak explosions.

Premixed combustion is much more difficult to model than non-premixed combustion. The reason for this is that premixed combustion usually occurs as a thin, propagating flame that is stretched and contorted by turbulence. For subsonic flows, the overall rate of propagation of the flame is determined by both the laminar flame speed and the turbulent eddies. The laminar flame speed is determined by the rate that species and heat diffuse upstream into the reactants and burn. To capture the laminar flame speed, the internal flame structure would need to be resolved, as well as the detailed chemical kinetics and molecular diffusion processes. Because practical laminar flame thicknesses are of the order of millimeters or smaller, resolution requirements are usually unaffordable.

The effect of turbulence is to wrinkle and stretch the propagating laminar flame sheet, increasing the sheet area and, in turn, the effective flame speed. The large turbulent eddies tend to wrinkle and corrugate the flame sheet, while the small turbulent eddies, if they are smaller than the laminar flame thickness, may penetrate the flame sheet and modify the laminar flame structure.

Non-premixed combustion, in comparison, can be greatly simplified to a mixing problem (see the mixture fraction approach in Introduction in the Theory Guide). The essence of premixed combustion modeling lies in capturing the turbulent flame speed, which is influenced by both the laminar flame speed and the turbulence.

In premixed flames, the fuel and oxidizer are intimately mixed before they enter the combustion device. Reaction then takes place in a combustion zone that separates unburnt reactants and burnt combustion products. Partially premixed flames exhibit the properties of both premixed and diffusion flames. They occur when an additional oxidizer or fuel stream enters a premixed system, or when a diffusion flame becomes lifted off the burner so that some premixing takes place prior to combustion.

Premixed and partially premixed flames can be modeled using ANSYS Fluent's finite-rate/eddy-dissipation formulation (see Modeling Species Transport and Finite-Rate Chemistry (p. 957)). If finite-rate chemical kinetic effects are important, the Laminar Finite-Rate model (see Direct Use of Finite-Rate Kinetics (no TCI) in the Theory Guide), the EDC model (see The Eddy-Dissipation-Concept (EDC) Model in the Theory Guide) or the composition PDF transport model (see Modeling a Composition PDF Transport Prob-

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

lem (p. 1113)) can be used. For information about ANSYS Fluent's partially premixed combustion model, see Modeling Partially Premixed Combustion (p. 1099). If the flame is perfectly premixed, so only one stream at one equivalence ratio enters the combustor, it is possible to use the premixed combustion model, as described in this chapter.

16.1.1. Limitations of the Premixed Combustion Model

The following limitations apply to the premixed combustion model:

- You must use the pressure-based solver. The premixed combustion model is not available with the density-based solver.
- The premixed combustion model is valid only for turbulent, subsonic flows. These types of flames are called deflagrations. Explosions, also called detonations, where the combustible mixture is ignited by the heat behind a shock wave, can be modeled with the finite-rate model using the density-based solver. See Modeling Species Transport and Finite-Rate Chemistry (p. 957) for information about the finite-rate model.
- The premixed combustion model cannot be used in conjunction with the pollutant (that is, soot and NOx) models. However, a perfectly premixed system can be modeled with the partially premixed model (see Modeling Partially Premixed Combustion (p. 1099)), which can be used with the pollutant models.
- You cannot use the premixed combustion model to simulate reacting discrete-phase particles, because these would result in a partially premixed system. Only inert particles can be used with the premixed combustion model.
- The G-Equation model can be used only with the unsteady solver because it tracks the flame front in time. However, RANS solutions, which tend to a steady-state, can be modeled by evolving them in time until the solution is stationary.

16.2. Using the Premixed Combustion Model

The procedure for setting up and solving a premixed combustion model is outlined below, and then described in detail. Remember that only the steps that are pertinent to premixed combustion modeling are shown here. For information about inputs related to other models that you are using in conjunction with the premixed combustion model, see the appropriate sections for those models.

1. Enable the premixed turbulent combustion model and set the related parameters.

   Note

   Make sure a turbulence model (other than the Spalart-Allmaras model) is selected before selecting a combustion model.

   Setup → **Models** → **Species** **Edit...**

2. Define the physical properties for the unburnt and burnt material in the domain.

   Setup → **Materials** → **Create/Edit...**

3. Set the value of the progress variable c at flow inlets and exits.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → **Boundary Conditions**

4. Initialize and patch the value of the progress variable.

   Solution → **Initialization** → **Patch...**

5. Solve the problem and perform postprocessing.

Important

If you are interested in computing the concentrations of individual species in the domain, you can use the partially premixed model described in Modeling Partially Premixed Combustion (p. 1099). Alternatively, compositions of the unburnt and burnt mixtures can be obtained from external analyses using equilibrium or kinetic calculations.

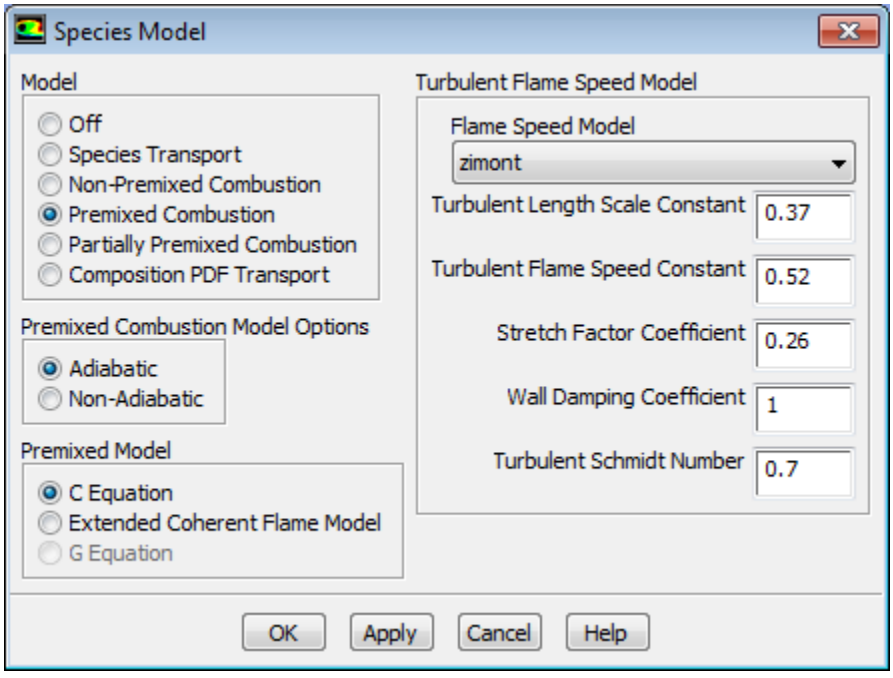
For additional information, see the following sections:
16.2.1. Enabling the Premixed Combustion Model
16.2.2. Choosing an Adiabatic or Non-Adiabatic Model

16.2.1. Enabling the Premixed Combustion Model

To enable the premixed combustion model, select **Premixed Combustion** under **Model** in the Species Model Dialog Box (p. 2216) (Figure 16.1: The Species Model Dialog Box for Premixed Combustion (p. 1091)).

Setup → **Models** → **Species** **Edit...**

Figure 16.1: The Species Model Dialog Box for Premixed Combustion



When you enable **Premixed Combustion**, the dialog box expands to show the relevant inputs.

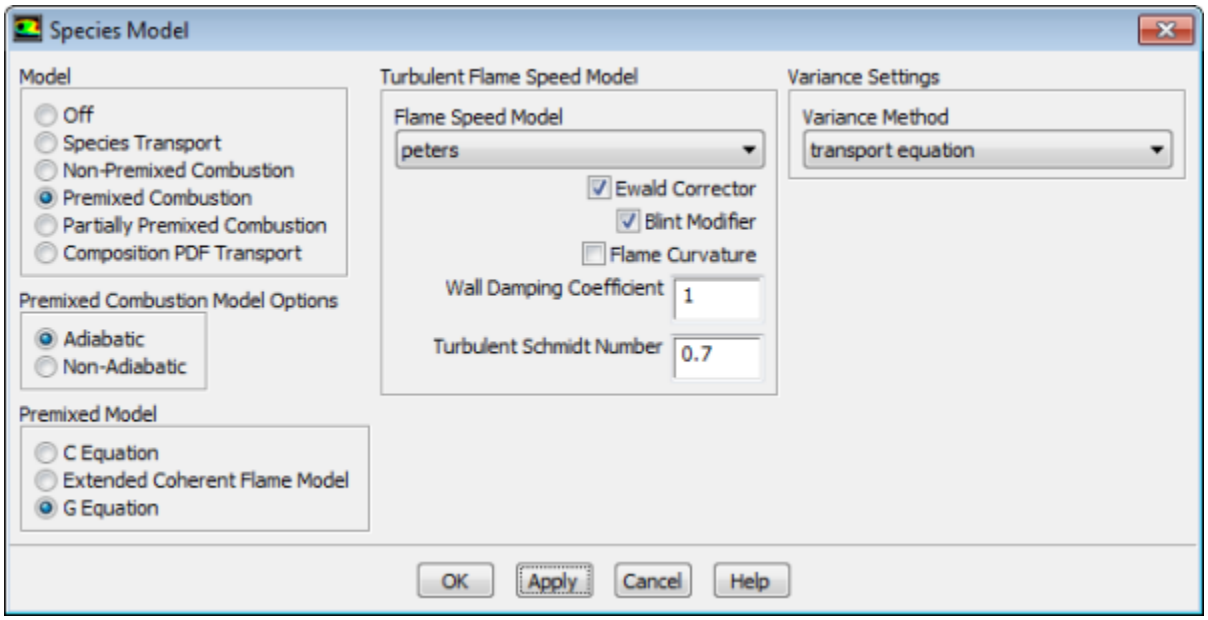
16.2.2. Choosing an Adiabatic or Non-Adiabatic Model

Under **Premixed Combustion Model Options** in the Species Model Dialog Box (p. 2216), choose either **Adiabatic** (the default) or **Non-Adiabatic**. This choice affects only the calculation method used to determine the temperature (either Equation 9.65 or Equation 9.66 in the Theory Guide).

16.3. Setting Up the C-Equation and G-Equation Models

When **C Equation** or **G Equation** is selected as the **Premixed Model**, you can then choose to use the **zimont** or **peters Flame Speed Model** (described in Turbulent Flame Speed Models). You can also modify a number of model constants, as described in Modifying the Constants for the Zimont Flame Speed Model (p. 1092) and Modifying the Constants for the Peters Flame Speed Model (p. 1093).

Figure 16.2: The Species Model Dialog Box for the G-Equation Model



For a non-adiabatic premixed combustion model, note that the value you specify for the **Turbulent Schmidt Number** will also be used as the Prandtl number for energy. (The **Energy Prandtl Number** will therefore not appear in the **Viscous Model** dialog box for non-adiabatic premixed combustion models.) These parameters control the level of diffusion for the progress variable and for energy. The progress variable is closely related to energy (because the flame progress results in heat release), so it is important that the transport equations use the same level of diffusion.

For additional information, see the following sections:

16.3.1. Modifying the Constants for the Zimont Flame Speed Model
16.3.2. Modifying the Constants for the Peters Flame Speed Model
16.3.3. Additional Options for the G-Equation Model
16.3.4. Defining Physical Properties for the Unburnt Mixture
16.3.5. Setting Boundary Conditions for the Progress Variable
16.3.6. Initializing the Progress Variable

16.3.1. Modifying the Constants for the Zimont Flame Speed Model

In general, you will not need to modify the constants used in the equations presented in C-Equation Model Theory in the Theory Guide. The default values are suitable for a wide range of premixed flames.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can set the **Turbulent Length Scale Constant** (C_D in Equation 9.11), **Turbulent Flame Speed Constant** (A in Equation 9.9), the **Stretch Factor Coefficient** (μ_{str} in Equation 9.16), the **Turbulent Schmidt Number** (Sc_t in Equation 9.1), and the **Wall Damping Coefficient** (α_w in Equation 9.19).

16.3.2. Modifying the Constants for the Peters Flame Speed Model

The Peters turbulent flame speed model constants, as described in Peters Flame Speed Model in the Theory Guide, are not available in the GUI because they are generally suitable for a wide range of premixed flames. They can, however, be accessed from the TUI.

The **Ewald Corrector** is enabled by default and described in Peters Flame Speed Model. The **Turbulent Schmidt Number** (Sc_t in Equation 9.1), and the **Wall Damping Coefficient** (α_w in Equation 9.19) are the same as those described for the Zimont model.

16.3.3. Additional Options for the G-Equation Model

When the G-Equation model is enabled, the **G Equation Settings** group box appears. You can select either the **transport equation** or **algebraic** option for the calculation of the flame distance variance. Consult Peters Flame Speed Model in the Theory Guide for the variance transport and algebraic equation expressions (Equation 9.6 and Equation 9.7). It is recommended that you use the **transport equation** option for RANS and the **algebraic** option for LES.

When **Flame Curvature Source** is enabled, the curvature source term in the G-Equation, which is the last term in Equation 9.4, is included. By default, this term is excluded.

16.3.4. Defining Physical Properties for the Unburnt Mixture

The fluid material in your domain should be assigned the properties of the unburnt mixture, including the thermal diffusivity (α in Equation 9.9 in the Theory Guide). α is defined as $k/\rho c_p$, and values at standard conditions can be found in combustion handbooks (for example, [55] (p. 2890)).

For both adiabatic and non-adiabatic combustion models, you will need to specify the **Laminar Flame Speed** (U_l in Equation 9.9 in the Theory Guide) as a material property, in the **Create/Edit Materials** dialog box. ANSYS Fluent will automatically select the **prepdf-polynomial** function for **Laminar Flame Speed**, indicating that the piecewise-linear polynomial function from the PDF look-up table will be used to compute the laminar flame speed for the partially-premixed model. You may also choose to enter a **constant** value, use a **user-defined** function, or apply the **metghalchi-keck-law** instead of a piecewise-linear polynomial function. See Laminar Flame Speed in the Theory Guide and Defining Physical Properties for the Unburnt Mixture (p. 1093) for information about setting the other properties for the unburnt material.

When using the Zimont turbulent flame speed model, if you want to include the flame stretch effect in your model, you will also need to specify a lower value than the default **Critical Rate of Strain** (g_{cr} in Equation 9.17 in the Theory Guide). As discussed in Flame Stretch Effect in the Theory Guide, g_{cr} is set to a very high value ($1\times10^8 s^{-1}$) by default, so no flame stretching occurs. To include flame stretching effects, you will need to adjust the **Critical Rate of Strain** based on experimental data for the burner. Because the flame stretching and flame extinction can influence the turbulent flame speed (as discussed in Flame Stretch Effect in the Theory Guide), a realistic value for the **Critical Rate of Strain** is required for accurate predictions. Typical values for CH_4 lean premixed combustion range from 3000 to 8000 s^{-1} [131] (p. 2894). Note that you can specify constant values or user-defined functions to define the

Laminar Flame Speed and **Critical Rate of Strain**. See the Fluent Customization Manual for details about user-defined functions.

For adiabatic models, you will also specify the **Adiabatic Burnt Temperature** (T_{ad} in Equation 9.65 in the Theory Guide), which is the temperature of the burnt products under adiabatic conditions. This temperature will be used to determine the linear variation of temperature in an adiabatic premixed combustion calculation. You can specify a constant value or use a user-defined function.

For non-adiabatic models, you will instead specify the **Heat of Combustion** per unit mass of fuel and the **Unburnt Fuel Mass Fraction** (H_{comb} and Y_{fuel} in Equation 9.67 in the Theory Guide). ANSYS Fluent will use these values to compute the heat losses or gains due to combustion, and include these losses/gains in the energy equation that it uses to calculate temperature. The **Heat of Combustion** can be specified only as a constant value, but you can specify a constant value or use a user-defined function for the **Unburnt Fuel Mass Fraction**.

To specify the density for a premixed combustion model, choose **premixed-combustion** in the **Density** drop-down list and set the **Adiabatic Unburnt Density** and **Adiabatic Unburnt Temperature** (T_u and ρ_u in Equation 9.68 in the Theory Guide). For adiabatic premixed models, your input for **Adiabatic Unburnt Temperature** (T_u) will also be used in Equation 9.65 in the Theory Guide to calculate the temperature.

The other properties specified for the unburnt mixture are viscosity, specific heat, thermal conductivity, and any other properties related to other models that are being used in conjunction with the premixed combustion model.

16.3.5. Setting Boundary Conditions for the Progress Variable

For premixed combustion models, you will need to set an additional boundary condition at flow inlets and exits: the progress variable, c. Valid inputs for the **Progress Variable** are as follows:

- $c=0$: unburnt mixture
- $c=1$: burnt mixture

16.3.6. Initializing the Progress Variable

Often, it is sufficient to initialize the progress variable c to 1 (burnt) everywhere and allow the unburnt ($c=0$) mixture entering the domain from the inlets to blow the flame back to the stabilizer. A better initialization is to patch an initial value of 0 (unburnt) upstream of the flame holder and a value of 1 (burnt) in the downstream region (after initializing the flow field in the **Solution Initialization** task page).

Solution → **Initialization** → **Patch...**

See Patching Values in Selected Cells (p. 1593) for details about patching values of solution variables.

16.4. Setting Up the Extended Coherent Flame Model

The Extended Coherent Flame Model (ECFM) solves a transport equation for the flame surface area density, denoted Σ, in addition to the reaction progress variable. Details about setting up the reaction progress variable boundary and initial conditions, and material properties, can be found in Setting Up the C-Equation and G-Equation Models (p. 1092). To use the ECFM, select **Extended Coherent Flame**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Model as the **Premixed Model**. A list of **Extended Coherent Flame Model Constants** will appear in the **Species Model** dialog box.

For additional information, see the following sections:
16.4.1. Modifying the ECFM Model Variant
16.4.2. Modifying the Constants for the ECFM Flame Speed Closure
16.4.3. Setting Boundary Conditions for the ECFM Transport
16.4.4. Initializing the Flame Area Density

16.4.1. Modifying the ECFM Model Variant

The source terms for Equation 9.27 in the Theory Guide are determined by which ECFM model variant is used. There are four ECFM model variants available. The default Veynante scheme is the recommended scheme, because it provides the best accuracy in most situations. If you want access to a different model variant, you must use the following text command:

`define` → `models` → `species` → `set-premixed-combustion`

You can then revise the model constants as necessary. See Closure for ECFM Source Terms in the Theory Guide for further details. If you are using LES turbulence modeling, see LES and ECFM as well.

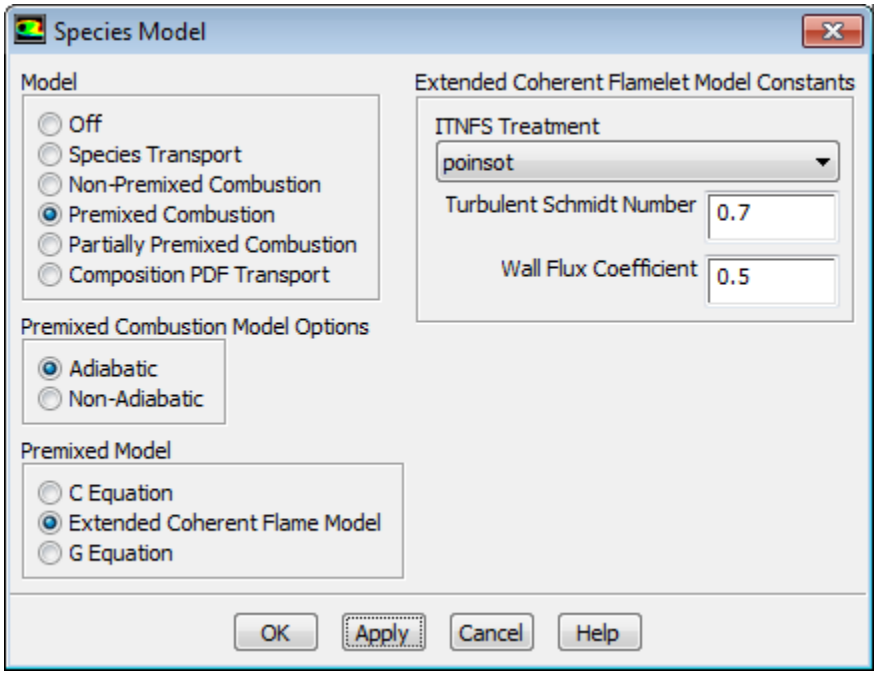
16.4.2. Modifying the Constants for the ECFM Flame Speed Closure

In the **Extended Coherent Flame Model Constants** group box of the **Species Model** dialog box, select the **ITNFS Treatment**. You have a choice of **constant-delta**, **meneveau**, **blint**, **poinsot**, or **constant**. The various ITNFS treatments are described in detail in Closure for ECFM Source Terms in the Theory Guide.

The treatment you select will determine which constants you can set. In all cases, you need to define the **Turbulent Schmidt Number** (Sc_t) and the **Wall Flux Coefficient**. If you selected the **constant-delta** treatment, you can set the **ITNFS Flame Thickness** (δ_l^0 in Equation 9.31 in the Theory Guide). If you selected the **constant** treatment, you can set the **ITNFS Value** (Γ_K in Equation 9.29 in the Theory Guide). Note that, in general, you will not need to modify the constants available in the **Extended Coherent Flame Model Constants** group box because the default values are suitable for a wide range of premixed flames.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.3: The Species Model Dialog Box for ECFM



16.4.3. Setting Boundary Conditions for the ECFM Transport

For the ECFM transport equation option for the premixed combustion models, you will need to set an additional boundary condition at flow inlets and exits: the flame area density, Σ. Valid inputs for the **Flame Area Density** are as follows:

- $\Sigma = 0$: no flame area (unburned)
- $\Sigma > 0$: burning with nonzero flame area

16.4.4. Initializing the Flame Area Density

Often, it is sufficient to initialize the flame area density Σ to 1 (laminar flame speed) everywhere and allow the unburnt mixture entering the domain from the inlets to develop. Another option for initialization is to patch an initial value of 0 (not burning) upstream of the flame holder and a value of 10 or higher (burning) in the downstream region (after initializing the flow field in the region).

See Patching Values in Selected Cells (p. 1593) for details about patching values of solution variables.

16.5. Postprocessing for Premixed Combustion Calculations

ANSYS Fluent provides several additional reporting options for premixed combustion calculations. You can generate graphical plots or alphanumeric reports of the following items:

- **Progress Variable**
- **Damkohler Number**
- **Stretch Factor**
- **Turbulent Flame Speed**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Static Temperature**
- **Product Formation Rate**
- **Laminar Flame Speed**
- **Critical Strain Rate**
- **Unburnt Fuel Mass Fraction**
- **Adiabatic Flame Temperature**

These variables are contained in the **Premixed Combustion...** category of the variable selection drop-down list that appears in postprocessing dialog boxes. See Field Function Definitions (p. 1931) for a complete list of flow variables, field functions, and their definitions. Displaying Graphics (p. 1747) and Reporting Alphanumeric Data (p. 1885) explain how to generate graphics displays and reports of data.

Note that **Static Temperature** and **Adiabatic Flame Temperature** will appear in the **Premixed Combustion...** category only for adiabatic premixed combustion calculations; for non-adiabatic calculations, **Static Temperature** will appear in the **Temperature...** category. **Unburnt Fuel Mass Fraction** will appear only for non-adiabatic models.

ANSYS Fluent also provides several additional reporting options for premixed combustion calculations with the ECFM model for flame speed closure. You can generate graphical plots or alphanumeric reports of the same items that are available with the premixed mode, and in addition:

- **Progress Variable Curvature**
- **Flame Area Density**
- **Net Flame Area Production**
- **Flame Area Production P1**
- **Flame Area Production P2**
- **Flame Area Production P3**
- **Flame Area Production P4**
- **Intermittent Turb Net Flame Stretch**
- **Flame Area Destruction**

These variables will also appear in the **Premixed Combustion...** category.

ANSYS Fluent also provides two additional reporting options for premixed combustion calculations with the G-Equation model:

- **Mean Distance from Flame**
- **Variance of Distance from Flame**

The **Variance of Distance from Flame** is the variance of the distance of the instantaneous flame from the mean flame position. These variables appear in the **Premixed Combustion...** category.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For additional information, see the following section:
16.5.1. Computing Species Concentrations

16.5.1. Computing Species Concentrations

If you know the composition of the unburnt and burnt mixtures in your model (that is, if you have performed separate ANSYS Fluent or external analyses of chemical equilibrium calculations or 1D premixed flames), you can compute the species concentrations in the domain using custom field functions:

- To determine the concentration of a species in the unburnt mixture, define the custom function $Y_u\left(1-c\right)$, where Y_u is the mass fraction for the species in the unburnt mixture (specified by you) and c is the value of the progress variable (computed by ANSYS Fluent).
- To determine the concentration of a species in the burnt mixture, define the custom function $Y_b c$, where Y_b is the mass fraction for the species in the burnt mixture (specified by you) and c is the value of the progress variable (computed by ANSYS Fluent).

See Custom Field Functions (p. 2003) for details about defining and using custom field functions.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 17: Modeling Partially Premixed Combustion

ANSYS Fluent provides a partially premixed combustion model that is based on the non-premixed combustion model described in Modeling Non-Premixed Combustion (p. 1025) and the premixed combustion model described in Modeling Premixed Combustion (p. 1089). For information about the theory behind the partially premixed combustion model, see Partially Premixed Combustion in the *Fluent Theory Guide*. Information about using the partially premixed combustion model is presented in the following sections:

17.1. Overview and Limitations
17.2. Using the Partially Premixed Combustion Model

17.1. Overview and Limitations

For additional information, see the following sections:

17.1.1. Overview
17.1.2. Limitations

17.1.1. Overview

Partially premixed combustion systems are premixed flames with non-uniform fuel-oxidizer mixtures (equivalence ratios). Such flames include premixed jets discharging into a quiescent atmosphere, lean premixed combustors with diffusion pilot flames and/or cooling air jets, and imperfectly mixed inlets.

The partially premixed model in ANSYS Fluent is a combination of the non-premixed model (Modeling Non-Premixed Combustion (p. 1025)) and the premixed model (Modeling Premixed Combustion (p. 1089)). The premixed reaction-progress variable, c, determines the position of the flame front. Behind the flame front ($c=1$), the mixture is burnt and the equilibrium or laminar flamelet mixture fraction solution is used. Ahead of the flame front ($c=0$), the species mass fractions, temperature, and density are calculated from the mixed but unburnt mixture fraction.

ANSYS Fluent offers two types of partially premixed combustion models, namely Thin Flamelet and Flamelet Generated Manifold (FGM).

The Thin Flamelet model assumes that the premixed flame front is infinitely thin, and that within the flame brush ($0<\overline{c}<1$), the thermochemistry is described by a linear combination of unburnt and burnt mixtures. The composition of the burnt mixture can be chemical equilibrium or steady diffusion laminar flamelets.

The FGM model assumes that the thermochemical states in a turbulent flame are similar to the states in a laminar flame, and parameterize these by mixture fraction and reaction progress. Within the laminar flame, reaction progress increases from $c=0$ in the unburnt reactants to $c=1$ in the burnt products, over a finite flame thickness. The FGM can be modeled with either premixed or diffusion laminar flames.

17.1.2. Limitations

- The underlying theory, assumptions, and limitations of the non-premixed and premixed models apply directly to the partially premixed model. In particular, the single-mixture-fraction approach is limited to two inlet

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

streams, which may be pure fuel, pure oxidizer, or a mixture of fuel and oxidizer. The two-mixture-fraction model extends the number of inlet streams to three, but incurs a major computational overhead. See Limitations of the Premixed Combustion Model (p. 1090) for additional limitations.

- The Flamelet Generated Manifold (FGM) model is limited to the C-Equation partially-premixed model.

17.2. Using the Partially Premixed Combustion Model

The procedure for setting up and solving a partially premixed combustion problem combines parts of the non-premixed combustion setup and the premixed combustion setup. An outline of the procedure is provided in Setup and Solution Procedure (p. 1100), along with information about where to look in the non-premixed and premixed combustion chapters for details. Inputs that are specific to the partially premixed combustion model are provided in this chapter.

Information is provided in the following sections:

17.2.1. Setup and Solution Procedure
17.2.2. Importing a Flamelet
17.2.3. Flamelet Generated Manifold
17.2.4. Calculating the Look-Up Tables
17.2.5. Standard Files for Flamelet Generated Manifold Modeling
17.2.6. Modifying the Unburnt Mixture Property Polynomials
17.2.7. Modeling In Cylinder Combustion

17.2.1. Setup and Solution Procedure

1. Read your mesh file into ANSYS Fluent and set up any other models you plan to use in conjunction with the partially premixed combustion model (turbulence, radiation, and so on).

2. Enable the partially premixed combustion model.

   a. Turn on the **Partially Premixed Combustion** model in the **Species Model** dialog box.

   Note

   Make sure a turbulence model (other than the Spalart-Allmaras model) is selected before selecting a combustion model.

   Setup → **Models** → **Species** **Edit...**

   b. If you select the **C Equation** as the **Premixed Model**, then you have the choice of selecting one of the following **State Relation** models in the **Chemistry** tab: **Chemical Equilibrim**, **Steady Diffusion Flamelet**, **Unsteady Diffusion Flamelet** (for steady-state simulations starting from a converged Steady Diffusion Flamelet solution), or **Flamelet Generated Manifold**.

   c. If you select the **Extended Coherent Flame Model**, then you can select **Chemical Equilibrim**, **Steady Diffusion Flamelet**, and **Unsteady Diffusion Flamelet**. You can modify the **Extended Coherent Flamelet Model Constants** in the **Premix** tab. See Modifying the Constants for the ECFM Flame Speed Closure (p. 1095) for details.

   d. If you select the **G Equation** as the **Premixed Model**, then you have the choice of selecting one of the following **State Relation** models in the **Chemistry** tab: **Chemical Equilibrim** or **Steady Diffusion Flamelet**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. In the **Chemistry** tab of the **Species Model** dialog box, select the appropriate **State Relation** based on the type of partially premixed model you want to simulate. In ANSYS Fluent, there are two types of partially premixed models, namely a thin-flamelet model (equilibrium and diffusion flamelet) and a finite-thickness flamelet model, which is a Flamelet Generated Manifold. You can refer to Non-Premixed Combustion in the *Fluent Theory Guide* to learn about diffusion flamelets or Partially Premixed Combustion Theory in the *Fluent Theory Guide* for FGMs. Also, you can refer to Overview in the *Fluent Theory Guide* for general information about the thin-flamelet and finite-thickness flamelet models.

4. If you select **Flamelet Generated Manifold**, you will also need to define whether the FGM is calculated from a premixed or a diffusion flamelet, as well as the **Turbulence-Chemistry Interaction** in the **Premix** tab. The three options (**Finite-Rate**, **Turbulent Flame Speed**, and **Finite-Rate/Turbulent Flame Speed**) are discussed in FGM Turbulent Closure in the *Fluent Theory Guide*.

   Note

   The **Finite-Rate/Turbulent Flame Speed** option uses the minimum of finite-rate and turbulent flame speed as discussed in the FGM Turbulent Closure in the *Fluent Theory Guide*. In the mixing layer regions, the adjacent cells could have different rate determining steps, which causes spurious species generation like generation of a product species. This can be minimized by selecting only **Turbulent Flame Speed** option.

5. In the **Flamelet** tab, you can modify the **Flamelet Parameters** as described in Flamelet Generated Manifold (p. 1103).

6. Generate a PDF look-up table. The table parameters are defined in Calculating the Look-Up Tables (p. 1105).

   Important

   If ANSYS Fluent warns you during the partially premixed properties calculation that any parameters are out of the range for the laminar flame speed function, you will need to modify the piecewise-linear points manually before saving the PDF file. See Modifying the Unburnt Mixture Property Polynomials (p. 1110) for details. Also, the calculation of the thermal diffusivity uses the thermal conductivity in the **Create/Edit Materials** dialog box. More accurate thermal diffusivity polynomials can be obtained by editing the thermal conductivity in the **Create/Edit Materials** dialog box and then clicking **Recalculate Properties** in the **Properties** tab.

7. Define the physical properties for the unburnt material in the domain.

   Setup → Materials

   ANSYS Fluent will automatically select the **prepdf-polynomial** function for **Laminar Flame Speed**, indicating that the piecewise-linear polynomial function from the PDF look-up table will be used to compute the laminar flame speed. You may also choose to enter a **constant** value, use a **user-defined** function, or apply the **metghalchi-keck-law** instead of a piecewise-linear polynomial function. See Laminar Flame Speed in the *Fluent Theory Guide* and Defining Physical Properties for the Unburnt Mixture (p. 1093) for information about setting the other properties for the unburnt material.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Set the values for the mean progress variable ($\overline{c}$) and the mean mixture fraction ($\overline{f}$) and its variance ($\overline{f'^2}$) at flow inlets and exits. (For problems that include a secondary stream, you will define boundary conditions for the mean secondary partial fraction and its variance as well.)

   Setup → **Boundary Conditions**

   See Defining Non-Premixed Boundary Conditions (p. 1079) for guidelines on setting mixture fraction and variance conditions, as well as thermal and velocity conditions at inlets.

   Important

   There are two ways to specify a premixed inlet boundary condition:

   a. If you defined the fuel composition in the **Boundary** tab to be the premixed inlet species, then you should set $\overline{f}=1$ and $\overline{c}=0$ in the boundary condition dialog boxes.

   b. If you set the fuel composition to pure fuel in the **Boundary** tab, you will need to set the correct equivalence ratio ($0<\overline{f}<1$) and $\overline{c}=0$ at your premixed inlet boundary condition.

   For example, if the premixed inlet of methane and air is at an equivalence ratio of 0.3, you can

   a. specify the mass fraction of the fuel composition of Y_{CH_4} = 0.017, Y_{O_2} = 0.236, and Y_{N_2} = 0.747 in the **Boundary** tab and $\overline{f}$ = 1 and $\overline{c}$ = 0 in the boundary condition dialog box.

   b. specify the mass fraction of the fuel composition of Y_{CH_4} = 1.0 in the **Boundary** tab and $\overline{f}$ = 0.017 and $\overline{c}$ = 0 in the boundary condition dialog box.

      Method (a) is preferred since it will have more points in the flame zone than method (b).

9. Initialize the value of the progress variable.

   Solution → **Initialization** → **Patch...**

   See Initializing the Progress Variable (p. 1094) for details.

10. Solve the problem and perform postprocessing.

    See Solving the Flow Problem (p. 1083) for guidelines about setting solution parameters. (These guidelines are for non-premixed combustion calculations, but they are relevant for partially premixed as well.)

17.2.2. Importing a Flamelet

To import an existing flamelet file

1. Select the **Import Flamelet** option in the **Chemistry** tab of the **Species Model** dialog box.

2. Click the **Import Flamelet File...** button. In The Select File Dialog Box (p. 19), select the file to be read in to ANSYS Fluent.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After you have completed this step, you can skip ahead to the **Table** tab of the **Species Model** dialog box (see Calculating the Look-Up Tables (p. 1105)).

17.2.3. Flamelet Generated Manifold

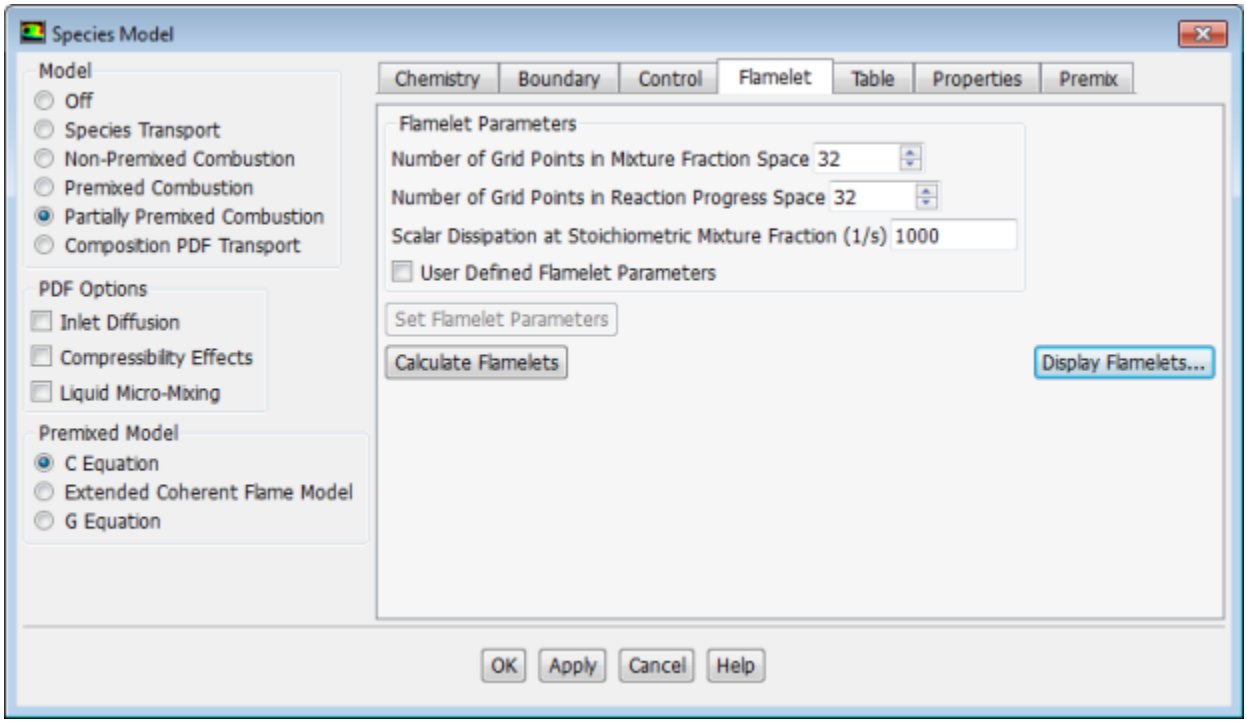
The ANSYS Fluent Flamelet Generated Manifold (FGM) model has several advantages over the thin-flame equilibrium or diffusion steady flamelet models, such as the ability to model flame quenching due to dilution. The manifold can be modeled with either a **Premixed Flamelet** or a **Diffusion Flamelet**.

To generate a flamelet file, go to the **Flamelet** tab of the **Species Model** dialog box (Figure 15.16: The Species Model Dialog Box (Flamelet Tab) (p. 1057)), where you will enter values for parameters of the flamelet.

Premixed Flamelet Generated Manifolds

For premixed Flamelet Generated Manifolds, details about specifying the Scalar Dissipation at Stoichiometric Mixture Fraction can be found in Premixed FGMs in the *Fluent Theory Guide*.

Figure 17.1: Premixed Flamelet Generated Manifolds (Flamelet Tab)



The **Flamelet Parameters** for premixed FGMs are as follows:

Number of Grid Points in Mixture Fraction Space

specifies the number of mixture fraction grid points distributed between the oxidizer ($f=0$) and the fuel ($f=1$). Increased resolution will provide greater accuracy, but since premixed flamelets are solved at all mixture fraction points, the solution time will increase linearly with this input.

Number of Grid Points in Reaction Progress Space

specifies the number of reaction progress grid points distributed between unburnt ($c=0$) and burnt ($c=1$) states. Increased resolution will provide greater accuracy, but since the premixed flamelet species and temperature are solved coupled and implicit in c space, the solution time and memory requirements increase greatly with the number of c grid points.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scalar Dissipation at Stoichiometric Mixture Fraction

specifies the premixed flamelet strain rate. The default of 1000/s is an approximate value of the scalar dissipation of a freely propagating (unstrained) premixed flame at stoichiometric mixture fraction (unity equivalence ratio).

User Defined Flamelet Parameters

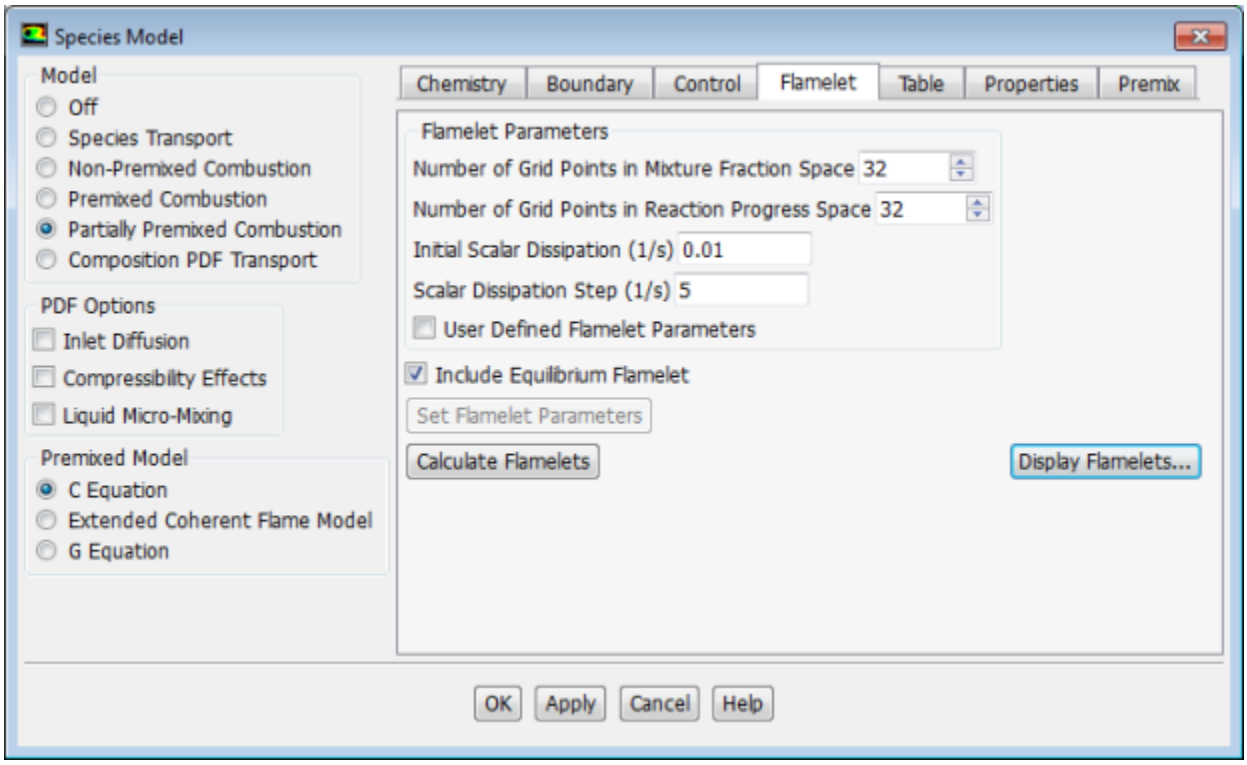
enables you to hook a user-defined function for scalar dissipation and mean mixture fraction (or progress variable) grid discretization

Click **Calculate Flamelets** to begin the premixed flamelet calculation.

Diffusion Flamelet Generated Manifolds

Generating diffusion Flamelet Generated Manifolds is similar to generating Steady Diffusion Flamelets, as described in Calculating the Flamelets (p. 1056).

Figure 17.2: Premixed Diffusion Flamelet Generated Manifolds (Flamelet Tab)



The parameters for the diffusion FGM flamelet are slightly different than those for the premixed FGM flamelet. The additional diffusion FGM flamelet parameters are as follows:

Initial Scalar Dissipation

is the scalar dissipation of the first flamelet in the library. This corresponds to χ_0 in Equation 8.52 in the *Fluent Theory Guide*.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scalar Dissipation Step

specifies the interval between scalar dissipation values (in s^{-1}) for which multiple flamelets will be calculated. This corresponds to $\Delta\chi$ in Equation 8.52 in the Theory Guide.

Note

The FGM model does not require the **Maximum Number of Flamelets** parameter. Instead, ANSYS Fluent increases the scalar dissipation until the flamelet extinguishes. Hence, the **Scalar Dissipation Step** should be set at approximately one twentieth of the counterflow diffusion flamelet extinction strain rate, so that about twenty flamelets are calculated. The final, extinguishing unsteady diffusion flamelet is used to model the manifold between the unburnt state (c=0) and the last steady diffusion flamelet at the highest scalar dissipation rate just before extinction.

Include Equilibrium Flamelet

determines how the progress variable (Equation 9.2 in the *Fluent Theory Guide*) is calculated. If this option is selected, ANSYS Fluent will compute the progress variable using equilibrium. Otherwise, the progress variable will be computed using the solution of flamelet generated with initial scalar dissipation as a denominator in Equation 9.2 in the *Fluent Theory Guide*.

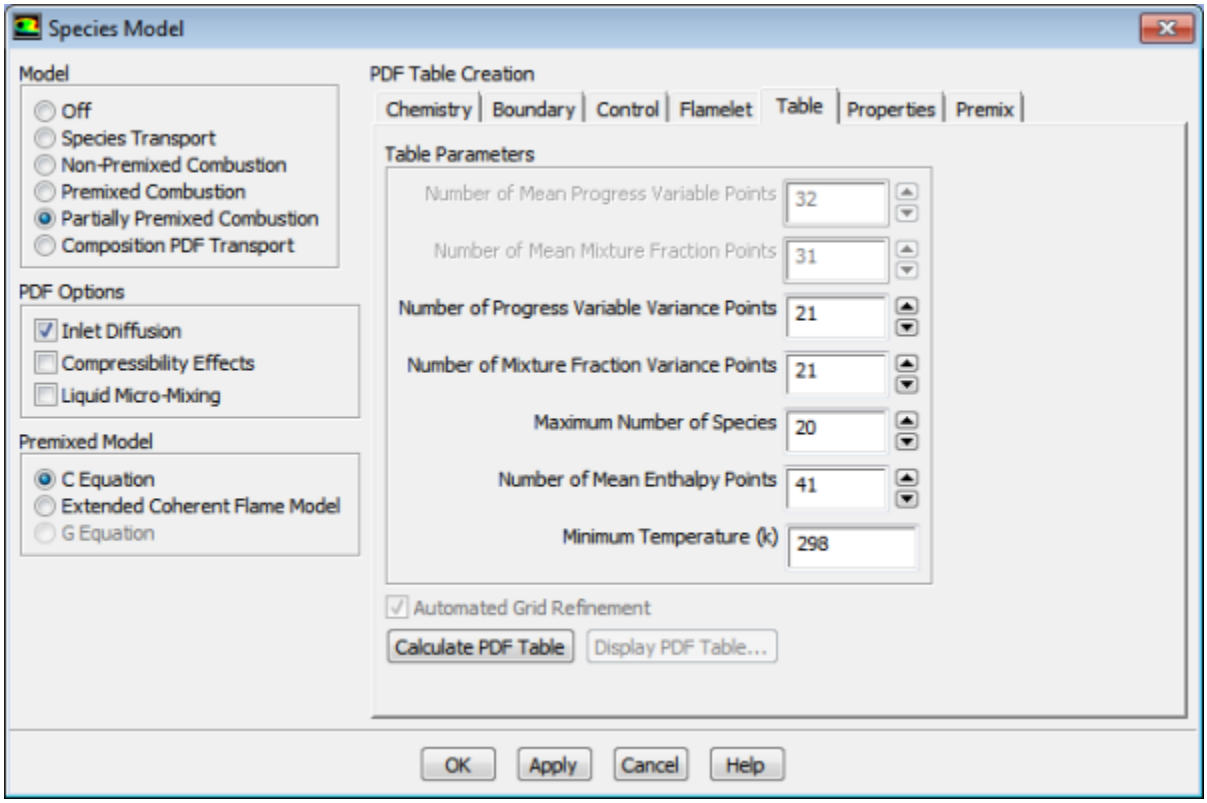
17.2.4. Calculating the Look-Up Tables

ANSYS Fluent requires additional inputs that are used in the creation of the look-up tables. Several of these inputs control the number of discrete values for which the look-up tables will be computed. These parameters are input in the **Table** tab of the **Species Model** dialog box.

For the example of modeling a partially-premixed flame with either the premixed or the diffusion FGM model, the **Table** tab of the **Species Model** dialog box is shown in Figure 17.3: The Species Model Dialog Box (Table Tab) (p. 1106).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 17.3: The Species Model Dialog Box (Table Tab)



The look-up table parameters are as follows:

Number of Mean Progress Variable Points

is the number of discrete values of $\overline{c}$ at which the look-up tables will be computed. This number is determined from the number of grid points in the flamelet.

Number of Progress Variable Variance Points

is the number of discrete values of $\overline{c'^2}$ at which the look-up tables will be computed. Lower resolution is acceptable because the variation along the $\overline{c'^2}$ axis is, in general, not as steep as the variation along the $\overline{c}$ axis of the look-up tables.

Maximum Number of Species

is the maximum number of species that will be included in the look-up tables. The maximum number of species that can be included is 200. ANSYS Fluent will automatically select the species with the largest mole fractions to include in the PDF table. Note that the PDF table values of density and specific heat are pre-calculated with all the species, and hence the convergence behavior of ANSYS Fluent will not be affected by the input for the **Maximum Number of Species**. Hence, to keep table sizes small, you should set the **Maximum Number of Species** to only include the species that you are interested in postprocessing.

Number of Mean Enthalpy Points

is the number of discrete values of enthalpy at which the look-up tables will be computed. This input is required only if you are modeling a non-adiabatic system. The number of points required will depend on the chemical system that you are considering, with more points required in high heat release systems (for example, hydrogen/oxygen flames).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Minimum Temperature
is used to determine the lowest temperature for which the look-up tables are generated. Your input should correspond to the minimum temperature expected in the domain (for example, an inlet or wall temperature). The minimum temperature should be set 10–20 K below the minimum system temperature. This option is available only if you are modeling a non-adiabatic system.

When you are satisfied with your inputs, click **Calculate PDF Table** to generate the look-up tables.

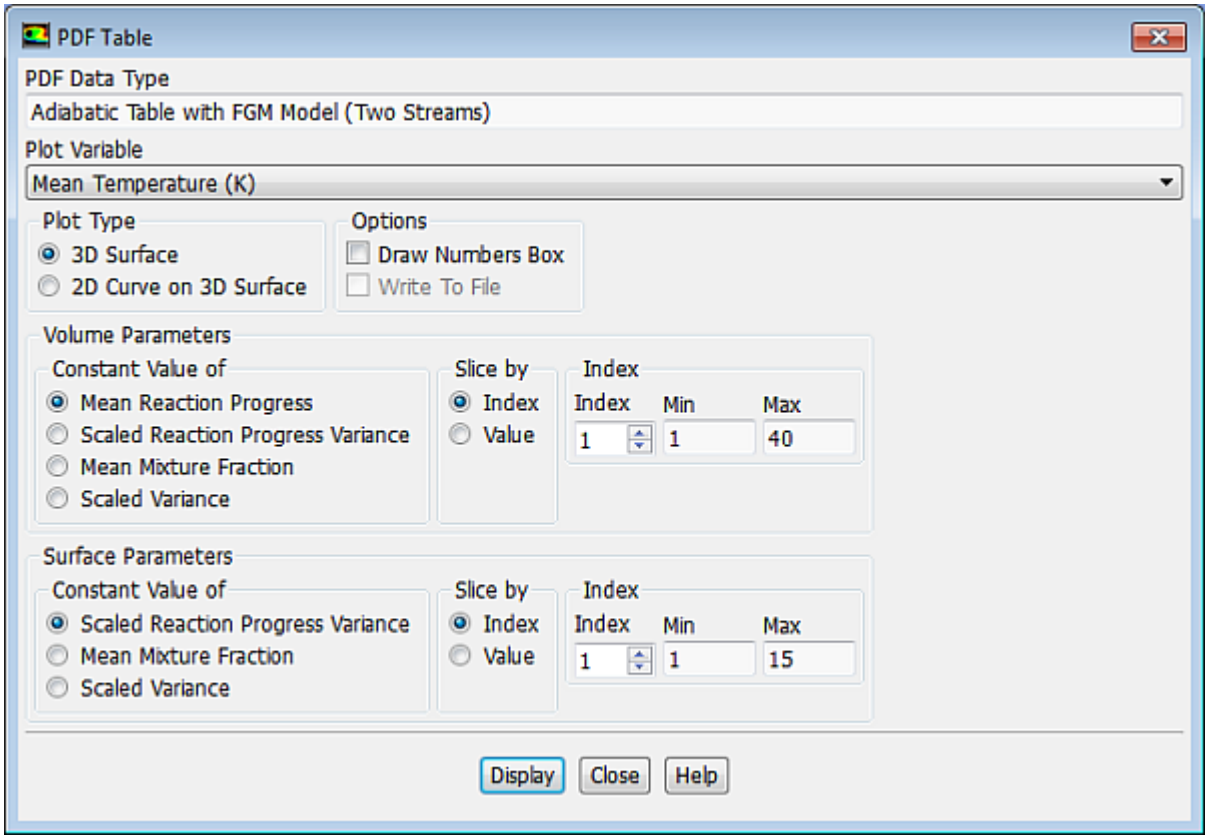
17.2.4.1. Postprocessing the Look-Up Tables with Flamelet Generated Manifolds

For the FGM model, the PDF table is four-dimensional for adiabatic cases and five-dimensional for non-adiabatic cases. For adiabatic PDF tables, the four independent variables are Mean mixture fraction, Mean reaction progress variable, Mixture fraction variance, and Progress variable variance. For non-adiabatic tables, the additional fifth independent variable is enthalpy. The FGM PDF tables can be post-processed using 2D curves and 3D surfaces by fixing two or more independent variables.

The procedure to display the lookup tables graphically is similar to that outlined in Postprocessing the Look-Up Table Data (p. 1068) with additional steps as follows.

- Adiabatic PDF Tables

Figure 17.4: The PDF Table Dialog Box (Adiabatic Case With FGM)



– 3D surfaces:

1. Under **Volume Parameters**, select the discrete independent parameter to be held constant and fix it by specifying either a slice **Index** number or a **Value**. (For more information, see Postprocessing the Look-Up Table Data (p. 1068).)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. In a similar manner, under **Surface Parameters**, select and fix the second discrete independent parameter and click **Plot**.

   The resulting plot is a 3D surface of the selected **Plot Variable** as a function of the remaining two independent variables that have not been fixed.

– 2D curves on 3D Surface:

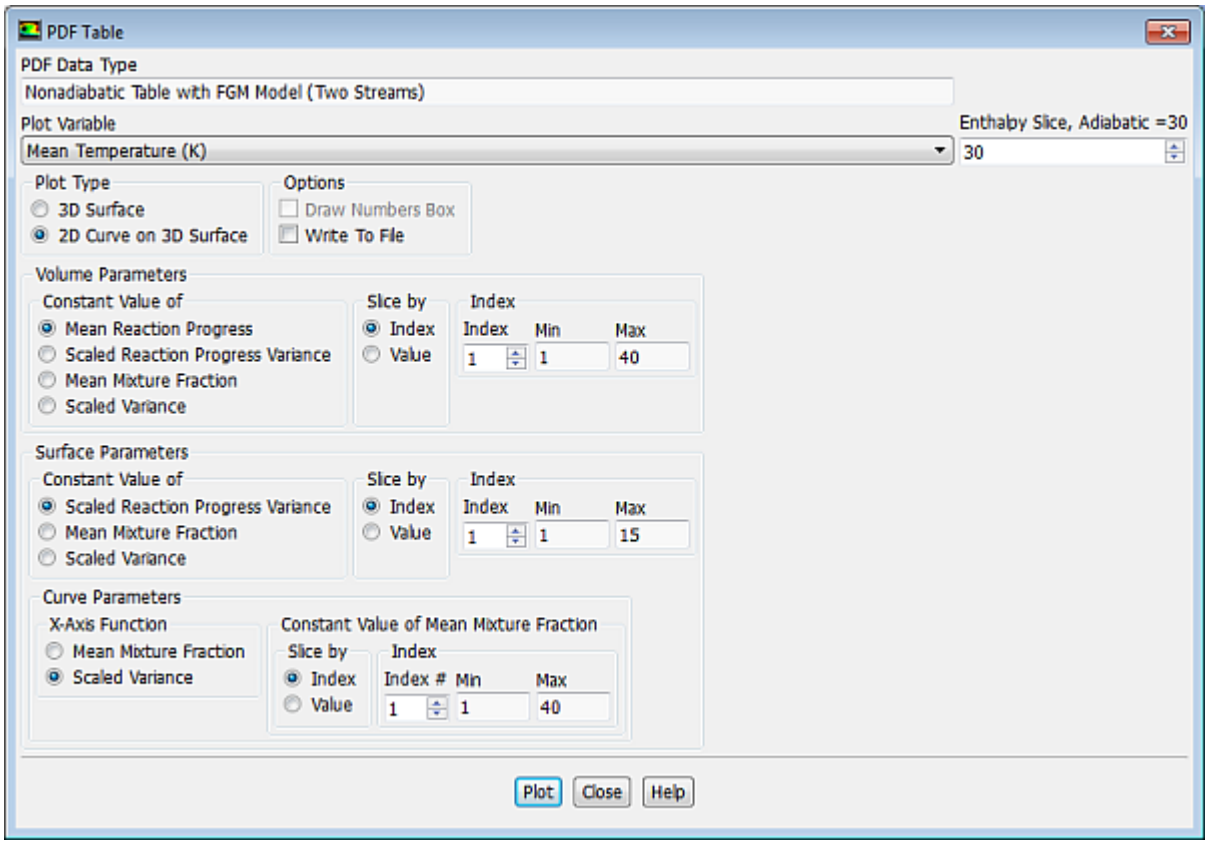
1. Under **Volume Parameters** and **Surface Parameters**, fix discrete independent parameters as described above.

2. Under the **Curve Parameters** group box, select the **X-Axis Function** against which the plot variable will be displayed, fix the remaining discrete independent parameter using controls in the **Constant Value of *`Independent Parameter`*** group box, and click **Plot**.

   The resulting plot is a 2D curve of the selected **Plot Variable** as a function of the remaining independent variable that has not been fixed.

• Non-Adiabatic PDF Tables

Figure 17.5: The PDF Table Dialog Box (Non-Adiabatic Case With FGM)



1. Fix the enthalpy slice level by specifying **Enthalpy Slice**.

   For non-adiabatic PDF tables, heat loss/gain is the fifth independent variable, which is expressed as enthalpy levels. By default, the enthalpy level is fixed at the adiabatic level, which is displayed in the **Enthalpy Slice, Adiabatic = *`slice number`*** label, as shown in Figure 17.5: The PDF Table Dialog Box (Non-Adiabatic Case With FGM) (p. 1108). Enthalpy slice numbers below the adiabatic

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

slice number are associated with heat loss, while enthalpy slice numbers above the adiabatic slice number are associated with enthalpy gain.

2. Depending on whether you want to display a 3D surface or a 2D Curve on 3D Surface, follow appropriate steps for Adiabatic PDF Tables described above.

17.2.5. Standard Files for Flamelet Generated Manifold Modeling

A standard flamelet file format can be used to read and write FGM flamelets. The data structure for the standard flamelet file format is based on keywords that precede each data section. If any of the keywords in your flamelet data file do not match the supported keywords, you will have to manually edit the file and change the keywords to one of the supported types. (The ANSYS Fluent flamelet filter is case-insensitive, so you need not worry about capitalization within the keywords.)

The following keywords are supported by the ANSYS Fluent filter:

- Header section: `HEADER`
- Main body section: `BODY`
- Number of species: `NUMOFSPECIES`
- Number of grid points: `GRIDPOINTS`
- Pressure: `PRESSURE`
- Temperature: `TEMPERATURE` and `TEMP`
- Mass fraction: `MASSFRACTION-`
- Mixture fraction: `Z`
- Mole fraction: `MOLEFRACTION-`
- Scalar dissipation for premixed flamelet: `PREMIX_CHI`
- Reaction progress: `REACTION_PROGRESS`
- Reaction progress source term: `PREMIX_YCDOT`

Note

Note that the FGM standard flamelet file format is identical for both premixed and diffusion flamelets. However, this format is different from the standard format for steady diffusion Laminar Flamelets, which is described in Calculating the Look-Up Tables (p. 1063).

17.2.5.1. Sample Standard FGM File

A sample FGM file in the standard FGM format is provided below. Note that not all species are listed in this file.

```
HEADER
PREMIX_STOICH_SCADIS   1.000000E+01
Z         0.000000E+00
NUMOFSPECIES  53
GRIDPOINTS  32
```

```
STOICH_Z    5.520020e-02
PRESSURE  1.013250E+05
BODY
REACTION_PROGRESS
0.000000000E+00   5.625000000E-02   1.125000000E-01   1.687500000E-01   2.250000000E-01
2.812500000E-01   3.375000000E-01   3.937500000E-01   4.500000000E-01   5.062500000E-01
5.625000000E-01   6.187500000E-01   6.750000000E-01   7.312500000E-01   7.875000000E-01
8.437500000E-01   9.000000000E-01   9.508132748E-01   9.604275079E-01   9.668740305E-01
9.718608224E-01   9.759835686E-01   9.795270729E-01   9.826516818E-01   9.854574013E-01
9.880111737E-01   9.903602004E-01   9.925391181E-01   9.945741609E-01   9.964857158E-01
9.982899681E-01   1.000000000E+00
TEMPERATURE
3.000000000E+02   3.000000063E+02   3.000000126E+02   3.000000189E+02   3.000000251E+02
3.000000314E+02   3.000000377E+02   3.000000440E+02   3.000000503E+02   3.000000566E+02
3.000000628E+02   3.000000691E+02   3.000000754E+02   3.000000817E+02   3.000000880E+02
3.000000943E+02   3.000001006E+02   3.000001062E+02   3.000001073E+02   3.000001080E+02
3.000001086E+02   3.000001090E+02   3.000001094E+02   3.000001098E+02   3.000001101E+02
3.000001104E+02   3.000001107E+02   3.000001109E+02   3.000001111E+02   3.000001113E+02
3.000001115E+02   3.000001117E+02
massfraction-h2
0.000000000E+00   1.095241207E-53   2.190482414E-53   3.285723620E-53   4.380964827E-53
5.476206034E-53   6.571447241E-53   7.666688448E-53   8.761929655E-53   9.857170861E-53
1.095241207E-52   1.204765328E-52   1.314289448E-52   1.423813569E-52   1.533337690E-52
1.642861810E-52   1.752385931E-52   1.851324229E-52   1.870044058E-52   1.882596053E-52
1.892305813E-52   1.900333194E-52   1.907232735E-52   1.913316647E-52   1.918779651E-52
1.923752089E-52   1.928325868E-52   1.932568429E-52   1.936530852E-52   1.940252832E-52
1.943765883E-52   1.947095479E-52
 .  .   .    .    .
 .  .   .    .    .
PREMIX_YCDOT
0.000000000E+00   0.000000000E+00   0.000000000E+00   0.000000000E+00   0.000000000E+00
0.000000000E+00   0.000000000E+00   0.000000000E+00   0.000000000E+00   0.000000000E+00
0.000000000E+00   0.000000000E+00   0.000000000E+00   0.000000000E+00   0.000000000E+00
0.000000000E+00   0.000000000E+00   0.000000000E+00   0.000000000E+00   0.000000000E+00
0.000000000E+00   0.000000000E+00   0.000000000E+00   0.000000000E+00   0.000000000E+00
0.000000000E+00   0.000000000E+00   0.000000000E+00   0.000000000E+00   0.000000000E+00
0.000000000E+00   0.000000000E+00
```

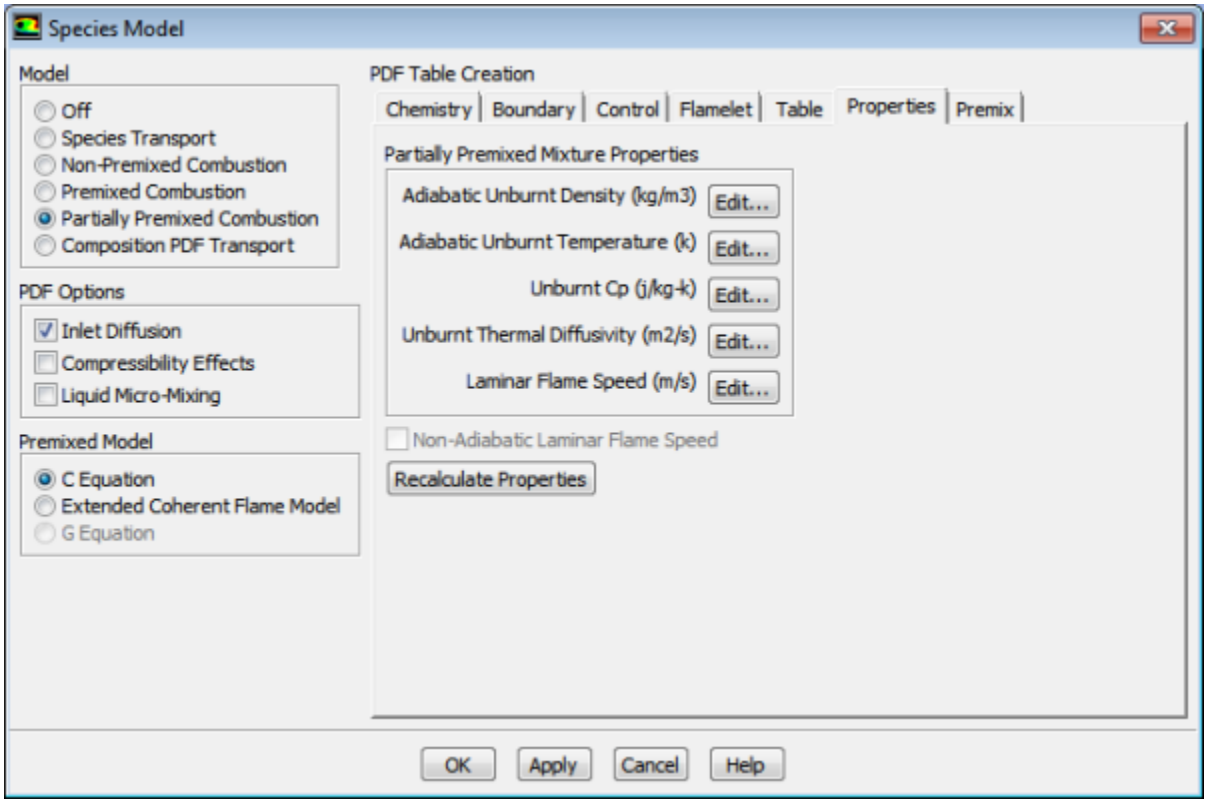
17.2.6. Modifying the Unburnt Mixture Property Polynomials

After building the PDF table, ANSYS Fluent automatically calculates the temperature, density, heat capacity, and thermal diffusivity of the unburnt mixture as polynomial functions of the mean mixture fraction, $\overline{f}$ (see Equation 10.20 in the *Fluent Theory Guide*). The laminar flame speed is automatically calculated as a piecewise-linear polynomial function of $\overline{f}$.

However, as outlined in Partially Premixed Combustion Theory in the *Fluent Theory Guide*, the laminar flame speed depends on details of the chemical kinetics and molecular transport properties, and is not calculated directly. Instead, curve fits are made to flame speeds determined from detailed simulations [38] (p. 2889). These fits are limited to a range of fuels (H_2, CH_4, C_2H_2, C_2H_4, C_2H_6, and C_3H_8), air as the oxidizer, equivalence ratios of the lean limit through unity, unburnt temperatures from 298 K to 800 K, and pressures from 1 bar to 40 bars. If your parameters fall outside this range, ANSYS Fluent will warn you when it computes the look-up table. In this case, you will need to modify the piecewise-linear points in the **Properties** tab of the **Species Model** dialog box (Figure 17.6: The Species Model Dialog Box (Properties Tab) (p. 1111)) before you save the PDF file.

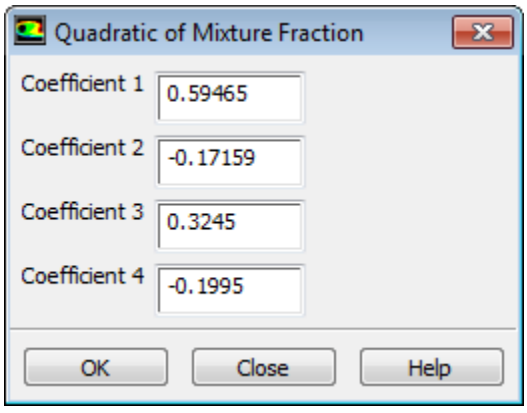
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 17.6: The Species Model Dialog Box (Properties Tab)



For each polynomial function of $\overline{f}$ under **Partially Premixed Mixture Properties** (**Adiabatic Unburnt Density**, **Adiabatic Unburnt Temperature**, **Unburnt Cp**, and **Unburnt Thermal Diffusivity**), you can specify values for **Coefficient 1**, **Coefficient 2**, **Coefficient 3**, and **Coefficient 4** (the polynomial coefficients in Equation 10.20 in the *Fluent Theory Guide*) in the appropriate **Quadratic of Mixture Fraction** dialog box (Figure 17.7: The Quadratic of Mixture Fraction Dialog Box (p. 1111)). To open this dialog box, click the appropriate **Edit...** button in the **Properties** tab.

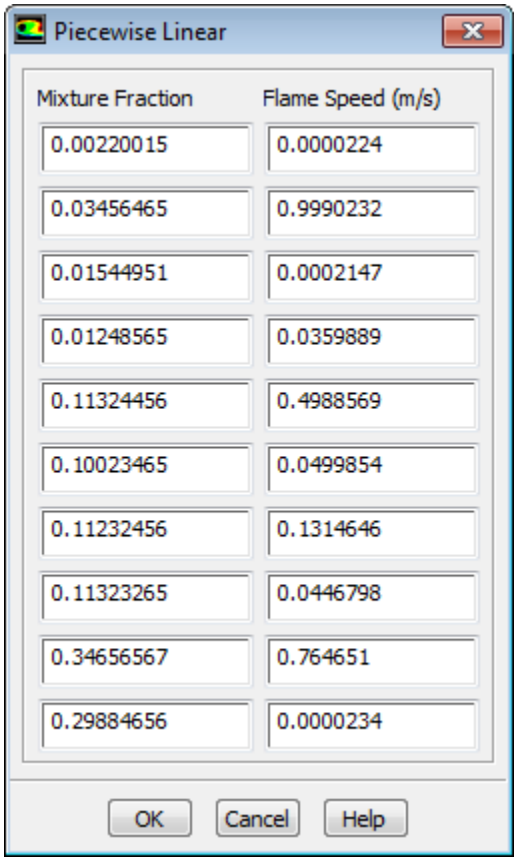
Figure 17.7: The Quadratic of Mixture Fraction Dialog Box



You can also specify the piecewise-linear **Mixture Fraction** and its corresponding **Laminar Flame Speed** for 10 different points in the **Piecewise Linear** dialog box (Figure 17.8: The Piecewise Linear Dialog Box (p. 1112)). The first set of values is the lower limit and the last set of values is the upper limit. Outside of either limit, the laminar flame speed is constant and equal to that limit. To open this dialog box, click the **Edit...** button next to **Laminar Flame Speed** in the **Properties** tab.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 17.8: The Piecewise Linear Dialog Box



Important

Note also that if you choose to use a user-defined function for the laminar flame speed in the Create/Edit Materials Dialog Box (p. 2302), the piecewise-linear fit becomes irrelevant.

If the secondary mixture fraction model is enabled, the unburnt properties are a function of both the mean and secondary mixture fractions. An additional column is added in the **Piecewise Linear** dialog box for the secondary mixture fraction unburnt polynomial coefficients.

Note that the **Non-Adiabatic Laminar Flame Speed** option when enabled includes the non-adiabatic effects on the laminar flame speed by tabulating the laminar speeds in the PDF table. See Laminar Flame Speed in the *Fluent Theory Guide*.

17.2.7. Modeling In Cylinder Combustion

Each of the partially premixed combustion models may be used for modeling in cylinder combustion. When modeling more than one cycle of such engines, you may choose to model trapped combustion products and exhaust gas recirculation (EGR) using the inert species model (see Setting Up the Inert Model (p. 1075)). A **Dynamic Mesh Event** has also been provided for calculating the inert composition, converting burned gases to inert and resetting the combustion process ready for the next cycle (see Resetting Inert EGR (p. 722) and Resetting Inert EGR (p. 1077)).

In the case of the G-Equation model, this is the only way to automatically model multiple in cylinder combustion cycles and take into account trapped burned gases remaining in the cylinder from one cycle to the next.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 18: Modeling a Composition PDF Transport Problem

ANSYS Fluent provides a composition PDF transport model for modeling finite-rate chemistry in turbulent flames. For information about the theory behind the composition PDF transport model, see Composition PDF Transport in the Theory Guide. Information about using this model is presented in the following sections:

18.1. Overview and Limitations
18.2. Steps for Using the Composition PDF Transport Model
18.3. Enabling the Lagrangian Composition PDF Transport Model
18.4. Enabling the Eulerian Composition PDF Transport Model
18.5. Initializing the Solution
18.6. Monitoring the Solution
18.7. Postprocessing for Lagrangian PDF Transport Calculations
18.8. Postprocessing for Eulerian PDF Transport Calculations

18.1. Overview and Limitations

The composition PDF transport model, like the Laminar Finite-Rate (see Direct Use of Finite-Rate Kinetics (no TCI) in the Theory Guide) and EDC model (see The Eddy-Dissipation-Concept (EDC) Model in the Theory Guide), should be used when you are interested in simulating finite-rate chemical kinetic effects in turbulent reacting flows. With an appropriate chemical mechanism, kinetically-controlled species such as CO and NOx, as well as flame extinction and ignition, can be predicted. PDF transport simulations are computationally expensive, and it is recommended that you start your modeling with small meshes, and preferably in 2D.

A limitation that applies to the composition PDF transport model is that you must use the pressure-based solver as the model is not available with the density-based solver.

ANSYS Fluent has two different discretizations of the composition PDF transport equation, namely Lagrangian and Eulerian. The Lagrangian method is more accurate than the Eulerian method, but requires significantly longer run time to converge.

18.2. Steps for Using the Composition PDF Transport Model

The procedure for setting up and solving a composition PDF transport problem is outlined below, and then described in detail. Remember that only steps that are pertinent to composition PDF transport modeling are shown here. For information about inputs related to other models that you are using in conjunction with the composition PDF transport model, see the appropriate sections for those models.

1. Read a CHEMKIN-formatted gas-phase mechanism file and the associated thermodynamic data file in the **CHEMKIN Mechanism** dialog box (see Importing a Volumetric Kinetic Mechanism in CHEMKIN Format (p. 967)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File → Import → CHEMKIN Mechanism...

Important

If your chemical mechanism is not in CHEMKIN format, you will have to enter the mechanism into ANSYS Fluent as described in Overview of User Inputs for Modeling Species Transport and Reactions (p. 958).

2. Enable a turbulence model.

   Setup → Models → Viscous Edit...

3. Enable the **Composition PDF Transport** model and set the related parameters. Refer to Enabling the Lagrangian Composition PDF Transport Model (p. 1115) and Enabling the Eulerian Composition PDF Transport Model (p. 1118) for further details.

   Setup → Models → Species Edit...

4. Check the material properties in the **Create/Edit Materials** dialog box and the reaction parameters in the **Reactions** dialog box. The default settings should be sufficient.

   Setup → Materials

5. Set the operating conditions, cell zone conditions, and boundary conditions.

   Setup → Cell Zone Conditions → Operating Conditions...

   Setup → Boundary Conditions

6. Check the solver settings.

   Solution → Methods

   Solution → Controls

   The default settings should be sufficient, although you should change the discretization to second-order once the solution has converged.

7. Initialize the solution. You may need to patch a high-temperature region to ignite the flame.

   Solution → Initialization → Initialize

   Solution → Initialization → Patch...

8. Run the solution.

   Solution → Run Calculation

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. Solve the problem and perform postprocessing.

Important

A good initial condition can reduce the solution time substantially. You should start from an existing solution calculated using the Laminar Finite-Rate, EDC model, non-premixed combustion model, or partially premixed combustion model. See Modeling Species Transport and Finite-Rate Chemistry (p. 957), Modeling Non-Premixed Combustion (p. 1025), and Modeling Partially Premixed Combustion (p. 1099) for further information on such simulations.

18.3. Enabling the Lagrangian Composition PDF Transport Model

To enable the composition PDF transport model, select **Composition PDF Transport** in the **Species Model** dialog box (Figure 18.1: The Species Model Dialog Box for Lagrangian Composition PDF Transport (p. 1116)).

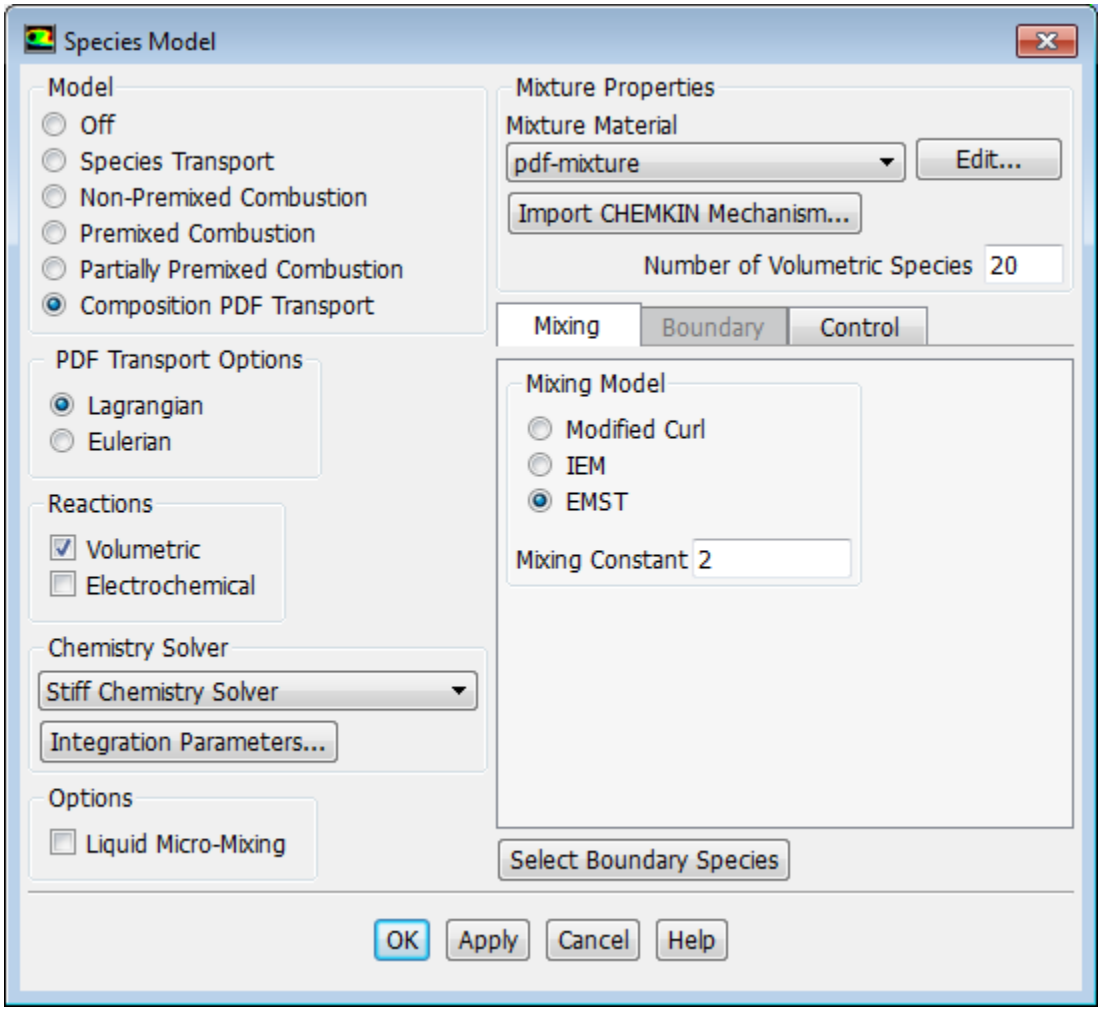
Setup → Models → Species Edit...

When you enable **Composition PDF Transport**, the dialog box will expand to show the relevant inputs.

1. Select **Lagrangian** under **PDF Transport Options**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 18.1: The Species Model Dialog Box for Lagrangian Composition PDF Transport



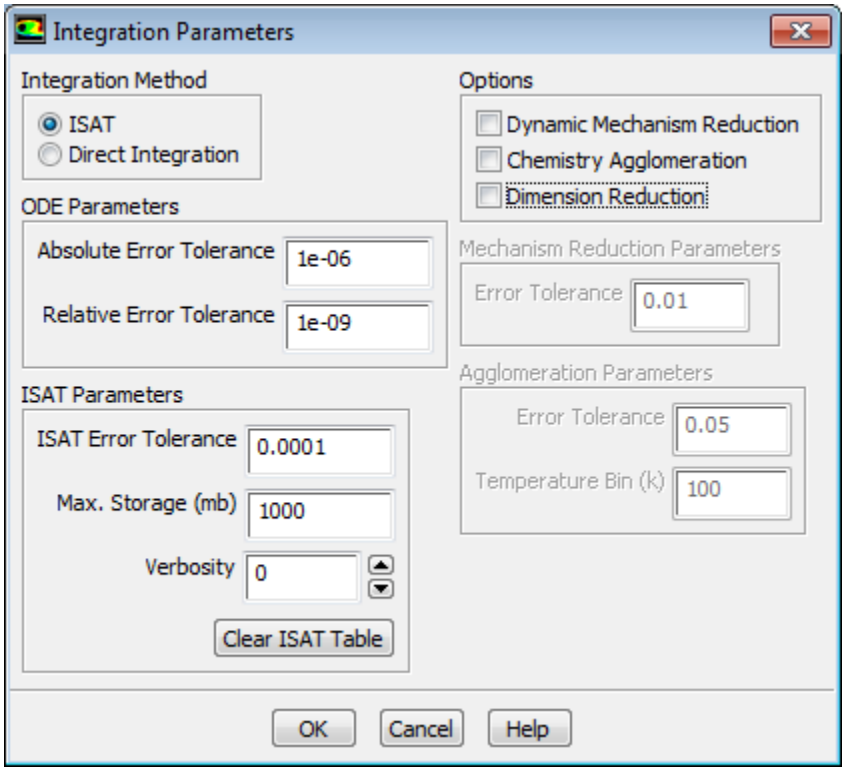
2. Enable **Volumetric** under **Reactions**.

   The **Composition Pdf Transport** model uses ANSYS Fluent stiff chemistry solver with ISAT enabled by default. See Using ISAT (p. 1128) for more details.

3. Click the **Integration Parameters...** button to open the Integration Parameters Dialog Box (p. 2235) (Figure 18.2: The Integration Parameters Dialog Box (p. 1117)). For additional information, see Using ISAT (p. 1128).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 18.2: The Integration Parameters Dialog Box



4. Enable **Liquid Micro-Mixing** to interpolate C_ϕ from turbulence models and scalar spectra, as noted in Liquid Reactions in the Theory Guide. This is applied to cases where reactions in liquids occur at low turbulence levels, among reactants with low diffusivities. Therefore, a default value of C_ϕ=2 may not be desirable, as it over-estimates the mixing rate.

5. In the **Mixing** tab, select **Modified Curl**, **IEM**, or **EMST** under **Mixing Model** and specify the value of the **Mixing Constant** (C_ϕ in Equation 11.6 in the Theory Guide). For more information about particle diffusion, see Particle Mixing in the Theory Guide.

> **Important**
>
> If the **Liquid Micro-Mixing** option is enabled, you cannot set the **Mixing Constant**.

6. You will not be specifying species boundary conditions in the **Boundary** tab. This is only applicable to the **Eulerian PDF Transport Option**.

7. In the **Control** tab, you will specify the Lagrangian PDF transport control parameters.

   Particles Per Cell
   sets the number of PDF particles per cell. Higher values of this parameter will reduce statistical error, but increase computational time.

   Local Time Stepping
   is available for steady-state simulations and can increase the convergence rate by taking maximum allowable time-steps on a cell-by-cell basis. (see Equation 11.4 of the Theory Guide). If **Local Time Stepping** is enabled, then you can specify the following parameters:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Convection #
specifies the particle convection number (see Δt_{conv} in Equation 11.4 in the Theory Guide).

Diffusion #
specifies the particle diffusion number (see Δt_{diff} in Equation 11.4).

Mixing #
specifies the particle mixing number (see Δt_{mix} in Equation 11.4).

18.4. Enabling the Eulerian Composition PDF Transport Model

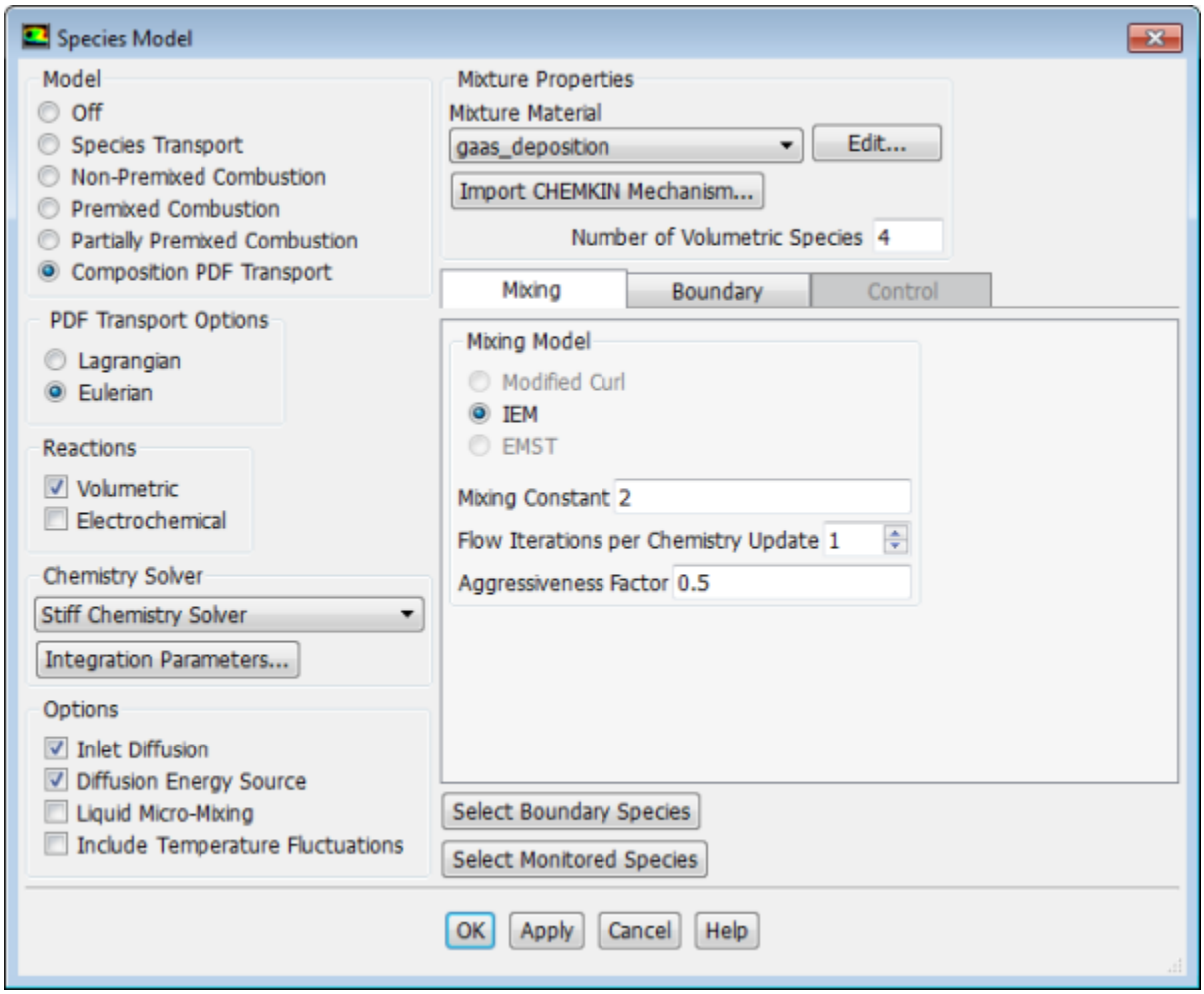
To enable the composition PDF transport model, select **Composition PDF Transport** in the **Species Model** dialog box (Figure 18.1: The Species Model Dialog Box for Lagrangian Composition PDF Transport (p. 1116)).

Setup → Models → Species Edit...

When you enable **Composition PDF Transport**, the dialog box will expand to show the relevant inputs.

1. Select **Eulerian** under **PDF Transport Options**.

Figure 18.3: The Species Model Dialog Box for Eulerian Composition PDF Transport



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Enable **Volumetric** under **Reactions**. The **Stiff Chemistry Solver** is disabled by default and should be enabled if the kinetic mechanism is numerically stiff.

3. Click the **Integration Parameters...** button to open the Integration Parameters Dialog Box (p. 2235) (Figure 18.2: The Integration Parameters Dialog Box (p. 1117)). See Using ISAT (p. 1128) for detailed information about this dialog box.

4. Enable **Inlet Diffusion** to include the diffusive transport of species through the inlets of your domain. Disable this option if the convective flux at one of the inlets is very small, resulting in mass loss by diffusion through that inlet.

5. Make sure that **Diffusion Energy Source** is enabled if you want to include species diffusion effects in the energy equation.

6. Enable **Liquid Micro-Mixing** if the fuel and oxidizer are liquids with low diffusivities (high Schmidt numbers). In this case, the **Mixing Constant** will be calculated according to Equation 11.13 in the Theory Guide.

7. By default, the Laminar (one mode) energy equation is solved, where temperature fluctuations are ignored. By enabling **Include Temperature Fluctuations**, the multi-mode energy equation will be solved as done for species.

8. In the **Mixing** tab, only the **IEM** mixing model is available for Eulerian PDF transport.

   a. Specify the value of the **Mixing Constant**. The default value is 2 for gas phase species.

   Important

   If the **Liquid Micro-Mixing** option is enabled, you cannot set the **Mixing Constant**.

   b. Enter the number of **Flow Iterations per Chemistry Update**. This is the frequency at which ANSYS Fluent will update the chemistry during the calculation. Increasing the number can reduce the computational expense of the chemistry calculations.

   c. Enter the **Aggressiveness Factor**. This is a numerical factor that controls the robustness and the convergence speed. This value ranges between 0 and 1, where 0 is the most robust, but results in the slowest convergence. The default value for the **Aggressiveness Factor** is 0.5.

9. In the **Boundary** tab, define the compositions of the fuel and oxidizer. You can select the boundary species to be displayed as described in Overview of the Problem Setup Procedure (p. 1026).

10. (**Eulerian** PDF Transport only) Optionally, you can select the monitored species as described in Overview of the Problem Setup Procedure (p. 1026).

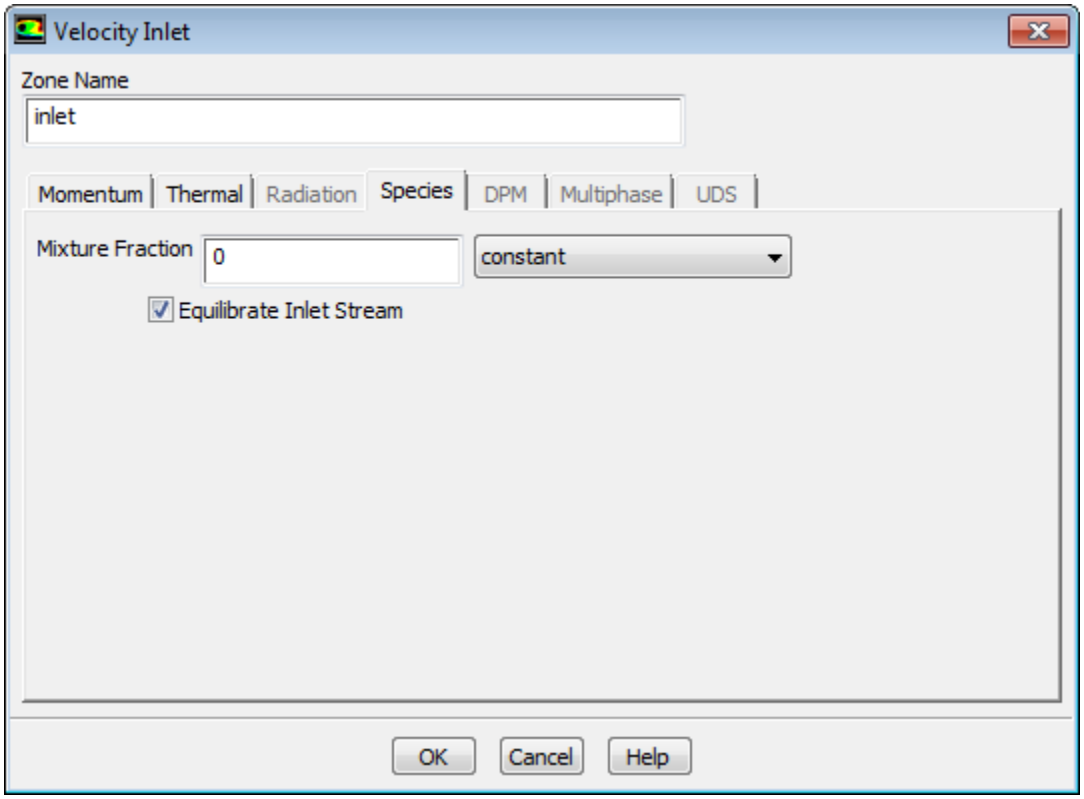
For additional information, see the following sections:
18.4.1. Defining Species Boundary Conditions

18.4.1. Defining Species Boundary Conditions

At flow inlets, specify the **Mixture Fraction** in the **Species** tab of the boundary condition inlet dialog boxes, as shown in Figure 18.4: The Velocity Inlet Dialog Box for Eulerian Composition PDF Transport (p. 1120). At outlet boundaries, similarly specify the **Backflow Mixture Fraction**. ANSYS Fluent always applies zero flux boundary conditions at walls for all species.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 18.4: The Velocity Inlet Dialog Box for Eulerian Composition PDF Transport



18.4.1.1. Equilibrating Inlet Streams

The **Equilibrate Inlet Stream** option in the **Species** tab of the boundary conditions dialog boxes will set the inlet compositions to their chemical equilibrium values. This option can be used to model pilot flames and exhaust gas recirculation.

Important

This option should not be enabled for pure fuel or oxidizer inlets.

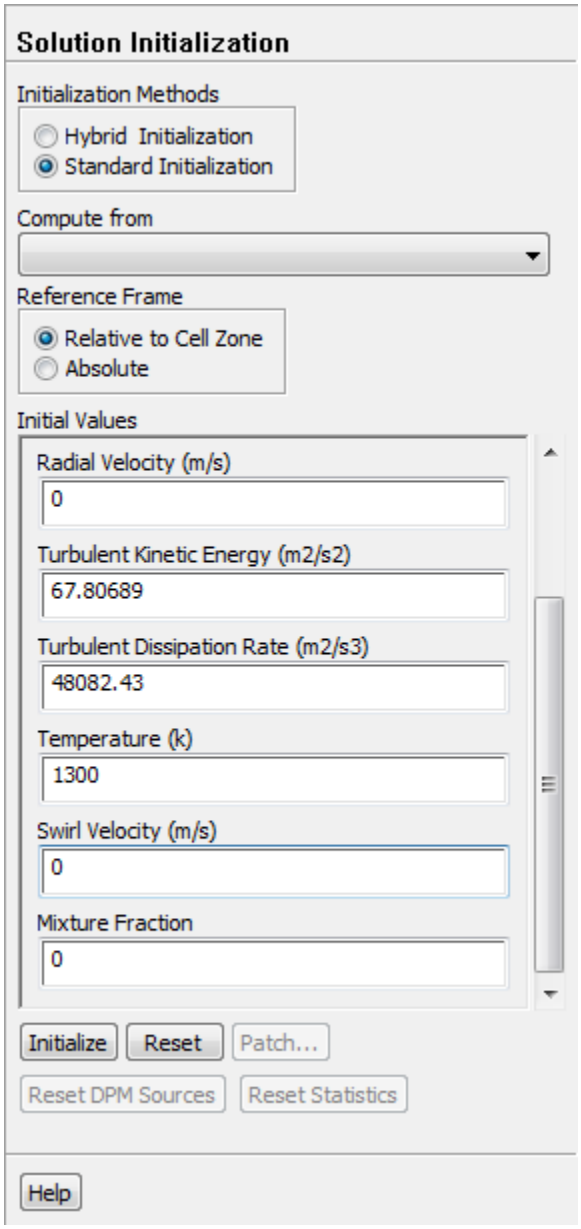
If you are using the Eulerian PDF transport model, specify the discretization and under-relaxation for the **Eulerian PDF** composition transport in the **Solution Controls** and **Solution Methods** task page.

18.5. Initializing the Solution

For the Eulerian PDF Transport model, the initialization variables are the mixture fraction and the temperature. The species mass fractions and enthalpy are calculated based on the fuel and oxidizer composition (specified in the **Species** dialog box) and the initialized mixture fraction and temperature.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 18.5: The Solution Initialization Task Page for Eulerian Composition PDF Transport

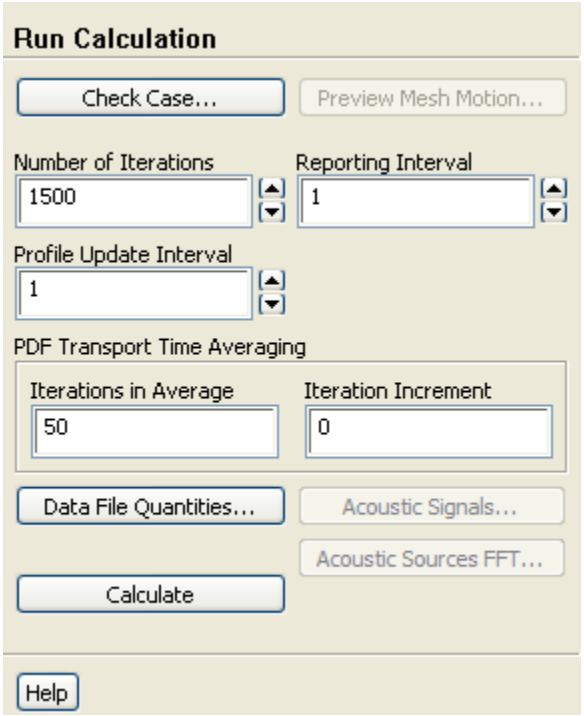


18.6. Monitoring the Solution

At low speeds, combustion couples to the fluid flow through density. The Lagrangian PDF transport algorithm has random fluctuations in the density field, which in turn causes fluctuations in the flow field. For steady-state flows, statistical fluctuations are decreased by averaging over a number of previous iterations in the **Run Calculation** task page (Figure 18.6: The Run Calculation Task Page for Composition PDF Transport (p. 1122)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 18.6: The Run Calculation Task Page for Composition PDF Transport



Averaging reduces statistical fluctuations and stabilizes the solution. However, ANSYS Fluent often indicates convergence of the flow field before the composition fields (temperatures and species) are converged. You should lower the default convergence criteria in the Residual Monitors Dialog Box (p. 2789), and always check that the **Total Heat Transfer Rate** in the Flux Reports Dialog Box (p. 2613) is balanced. It is also recommended that you monitor temperature/species on outlet boundaries and ensure that these are steady.

The Lagrangian PDF method has the additional solution controls: **Iterations in Average** and **Iteration Increment**. By increasing the **Iterations in Average**, fluctuations are smoothed out and residuals level off at smaller values. However, the composition PDF method requires a larger number of iterations to reach steady-state. It is recommended that you use the default of 50 **Iterations in Average** until the steady-state solution is obtained. Then, to gradually decrease the residuals, increase the **Iterations in Average** by setting a **Iteration Increment** to a value from 0 to 1 (the value 0.2 is recommended). Subsequent iterations will increase the **Iterations in Average** by the **Iteration Increment**.

For additional information, see the following sections:

18.6.1. Running Unsteady Composition PDF Transport Simulations
18.6.2. Running Compressible Lagrangian PDF Transport Simulations
18.6.3. Running Lagrangian PDF Transport Simulations with Conjugate Heat Transfer

18.6.1. Running Unsteady Composition PDF Transport Simulations

For unsteady Lagrangian composition PDF transport simulations, a fractional step scheme is employed where the PDF particles are advanced over the time step, and then the flow is advanced over the time step. Unlike steady-state simulations, composition statistics are not averaged over iterations, and to reduce statistical error you should increase the number of particles per cell in the **Solution Monitors** dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For low speed flows, the thermo-chemistry couples to the flow through density. Statistical errors in the calculation of density may cause convergence difficulties between time step iterations. If you experience this, increase the number of PDF particles per cell, or decrease the density under-relaxation.

18.6.2. Running Compressible Lagrangian PDF Transport Simulations

Compressibility is included when **ideal-gas** is selected as the density method in the Create/Edit Materials Dialog Box (p. 2302). For such flows, particle internal energy is increased by $p\Delta v$ over the time step Δt, where p is the cell pressure and Δv is the change in the particle specific volume over the time step.

18.6.3. Running Lagrangian PDF Transport Simulations with Conjugate Heat Transfer

When solid zones are present in the simulation, ANSYS Fluent solves the energy equation in the turbulent flow zones by the Lagrangian Monte Carlo particle method, and the energy equation in the solid zones by the finite-volume method.

18.7. Postprocessing for Lagrangian PDF Transport Calculations

For additional information, see the following sections:

18.7.1. Reporting Options
18.7.2. Particle Tracking Options

18.7.1. Reporting Options

ANSYS Fluent provides several reporting options for the Lagrangian composition PDF transport calculations. You can generate graphical plots or alphanumeric reports of the following items:

- **Static Temperature**
- **Mean Static Temperature**
- **RMSE Static Temperature**
- **Mass fraction of species-n**
- **Mean species-n Mass Fraction**
- **RMS species-n Mass Fraction**

The instantaneous composition (**Static Temperature** and **Mass fraction of species-n**) in a cell are calculated as,

$$\phi_{instant} = \frac{\sum_{i=1}^{N_c} \phi_p m_p}{\sum_{i=1}^{N_c} m_p} \tag{18.1}$$

where

$\phi_{instant}$ = instantaneous cell species mass fraction or temperature at the present iteration

N_c = number of particles in the cell

ϕ_p = particle mass fraction or temperature

m_p = particle mass

Mean and root-mean-square-error (RMSE) temperatures are calculated in ANSYS Fluent by averaging instantaneous temperatures over a user-specified number of previous iterations (see Monitoring the Solution (p. 1121)).

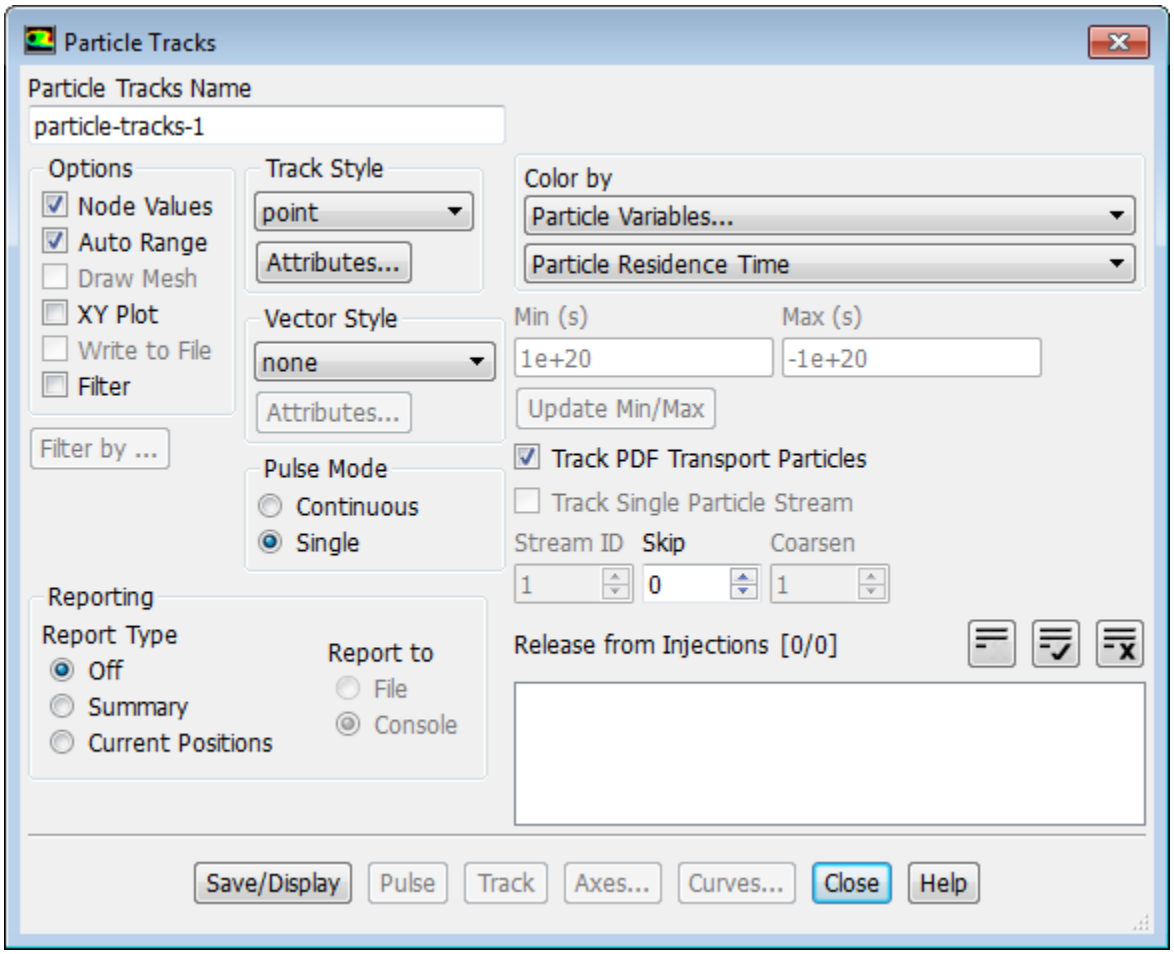
Note that for steady-state simulations, instantaneous temperatures and species represent a Monte Carlo realization and are as such unphysical. Mean and RMSE quantities are much more useful.

18.7.2. Particle Tracking Options

When you have enabled the Lagrangian composition PDF transport model, you can display the trajectories of the PDF particles using the Particle Tracks Dialog Box (p. 2761) (Figure 18.7: The Particle Tracks Dialog Box for Tracking PDF Particles (p. 1124)).

Results → Graphics → Particle Tracks → New...

Figure 18.7: The Particle Tracks Dialog Box for Tracking PDF Particles



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select the **Track PDF Transport Particles** option to enable PDF particle tracking. To speed up the plotting process, you can specify a value n for **Skip**, which will plot only every n^{th} particle. For details about setting other parameters in the **Particle Tracks** dialog box, see Displaying of Trajectories (p. 1330).

When you have finished setting parameters, click **Display** to display the particle trajectories in the graphics window.

18.8. Postprocessing for Eulerian PDF Transport Calculations

For additional information, see the following sections:
18.8.1. Reporting Options

18.8.1. Reporting Options

To postprocess the Eulerian composition PDF transport model, the following variables are available for each mode:

- **Mixture fraction**
- **Mass fraction of species in mode n**
- **Temperature of mode n**
- **Sensible Enthalpy of mode n**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 19: Using Chemistry Acceleration

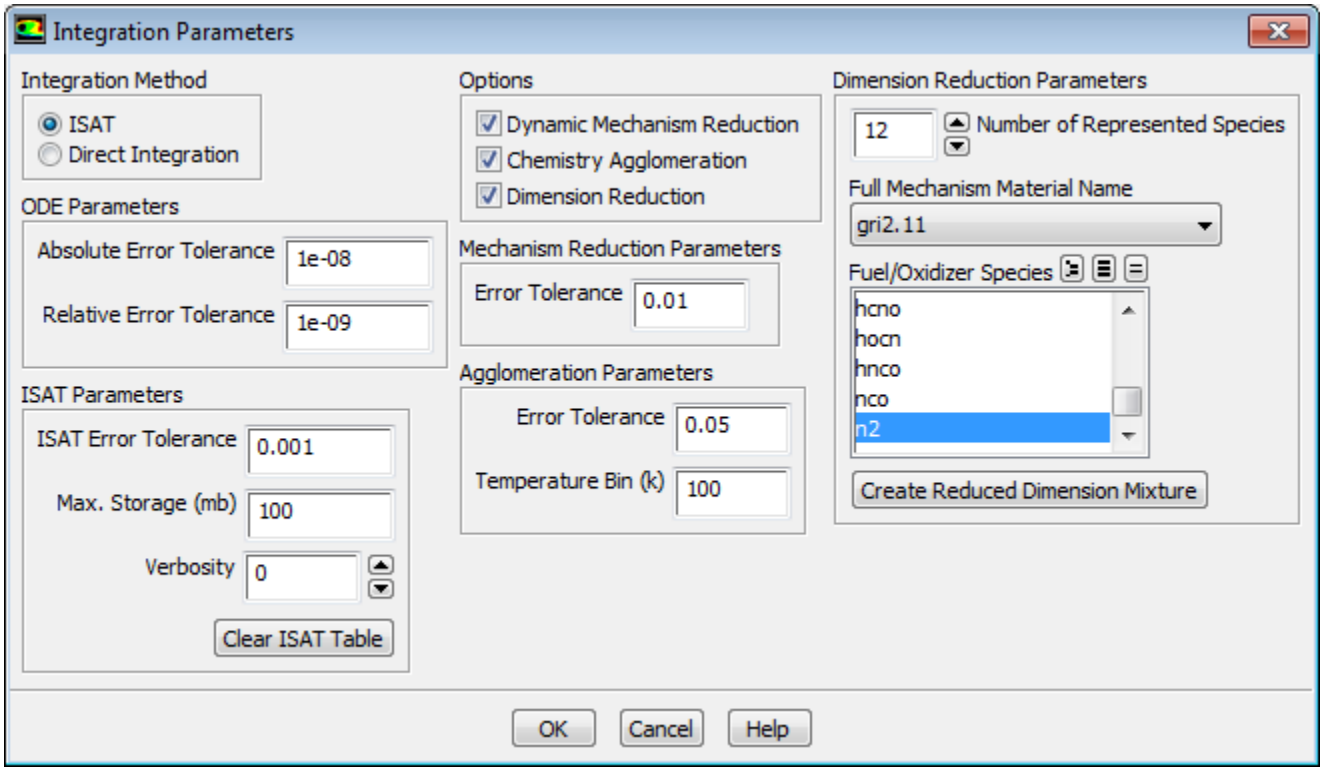
ANSYS Fluent can model detailed chemical kinetics in laminar and turbulent flames. Laminar flames are modeled with the **Laminar Finite-Rate** option, while four turbulence-chemistry interaction models are available for turbulent flames (**Laminar Finite-Rate**, **Eddy-Dissipation Concept**, **Lagrangian PDF Transport**, and **Eulerian PDF Transport**). Detailed chemical mechanisms are invariably numerically stiff and compute-intensive. ANSYS Fluent provides following methods to accelerate these computations:

- ISAT
- Dynamic Mechanism Reduction (DMR)
- Chemistry Agglomeration
- Dimension Reduction
- Dynamic Cell Clustering (DCC) (available with the transient ANSYS CHEMKIN-CFD solver)

All of the methods are enabled or disabled in the **Integration Parameters** dialog box, accessed from the **Species Model** dialog box by clicking the **Integration Parameters...** button in the **Reactions** group box:

Setup → Models → Species Edit...

Figure 19.1: The Integration Parameters Dialog Box Displaying Dimension Reduction



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

All of the acceleration methods induce some accuracy loss, and the controlling parameters should be carefully adjusted to ensure that this inaccuracy is acceptable, see Chemistry Acceleration in the *Fluent Theory Guide*. In most cases, applying ISAT and Dynamic Mechanism Reduction (DMR) together should give the best performance. Further information about using the chemistry acceleration models is presented in the following sections:

19.1. Using ISAT
19.2. Using Dynamic Mechanism Reduction
19.3. Using Chemistry Agglomeration
19.4. Dimension Reduction
19.5. Using Dynamic Cell Clustering

19.1. Using ISAT

In-Situ Adaptive Tabulation (ISAT) is a storage-retrieval method that constructs a chemistry table at run time (in-situ) with a user-specified interpolation accuracy (adaptive tabulation). ISAT can be used with all Chemistry solvers except the **None - Explicit Source** solver option.

For ODE solvers (Stiff Chemistry and CHEMKIN-CFD), ANSYS Fluent uses two error tolerances (under **ODE Parameters**):

- **Absolute Error Tolerance**: by default, set to 10^{-8}.
- **Relative Error Tolerance**: by default, set to 10^{-9} for the Stiff Chemistry solver and 10^{-4} for the CHEMKIN-CFD solver.

The default values should be sufficient for most applications, although these tolerances may need to be decreased for some cases such as ignition. For problems in which the accuracy of the chemistry integrations is crucial, it may be useful to test the accuracy of the error tolerances in simple zero-dimensional and one-dimensional test simulations with parameters comparable to those in the full simulation.

If you want to use chemistry agglomeration together with ISAT, enable **Chemistry Agglomeration** in the **Integration Parameters** dialog box and specify the chemistry agglomeration **Error Tolerance** (ε_{tol}). More information can be found in Using Chemistry Agglomeration (p. 1136).

For additional information, see the following sections:

19.1.1. ISAT Parameters
19.1.2. Monitoring ISAT
19.1.3. Using ISAT Efficiently
19.1.4. Reading and Writing ISAT Tables

19.1.1. ISAT Parameters

If you have selected **ISAT** under **Integration Method**, you will then be able to set additional ISAT parameters. The numerical error in the ISAT table is controlled by the **ISAT Error Tolerance** under **ISAT Parameters**. The default **ISAT Error Tolerance** of 0.001 may be sufficiently accurate for temperature and major species, but will most likely need to be decreased to get accurate minor species and pollutant predictions. For steady-state simulations, it may help to start with a high error tolerance during the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

initial iterations towards a converged solution. A larger error tolerance results in smaller tables and quicker run times, but greater error.

Important

After your steady simulation is converged, you should always decrease the **ISAT Error Tolerance** and perform further iterations until the species that you are interested in are unchanged.

The **Max. Storage** is the maximum RAM used by the ISAT table, and has a default value of 100 MB. Generally, you do not need to set it to a value larger than 500 MB. The value of **Verbosity** allows you to monitor ISAT performance in different levels of detail. See Monitoring ISAT (p. 1129) for details about this parameter.

To purge the ISAT table, click the **Clear ISAT Table** button. See Using ISAT Efficiently (p. 1130) for more details.

19.1.2. Monitoring ISAT

You can monitor ISAT performance by setting the **Verbosity** in the **Integration Parameters** dialog box. For a **Verbosity** of 1 or 2, ANSYS Fluent writes the following information periodically to a file named `case-file-name_stats.out`:

- total number of *queries*
- total number of queries resulting in *retrieves*
- total number of queries resulting in *grows*
- total number of queries resulting in *adds*
- total number of queries resulting in *direct integrations*
- cumulative CPU seconds in ISAT
- cumulative CPU seconds outside ISAT
- cumulative wall-clock time in seconds (that is, total CPU time in ISAT plus total CPU time out of ISAT plus CPU idle time)

The ISAT **Verbosity** option of 2 is for expert users who are familiar with ISATAB v5.0 [88] (p. 2891). ANSYS Fluent writes out the following files for **Verbosity** = 2:

- `tablename_stats.out`, as described above
- `tablename_ODE_accuracy.out` reports the accuracy of the ODE integrations. For every new ISAT table entry, if the maximum absolute error in temperature or species is greater than any previous error, a line is written to this file. This line consists of the total number of ODE integrations performed up to this time, the maximum absolute species error, the absolute temperature error, the initial temperature and the time step.
- `tablename_ODE_diagnostic.out` prints diagnostics from the ODE solver
- `tablename_ODE_warning.out` prints warnings from the ODE solver

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Initially, the table name is equal by default to the current case name, and is changed as the table is written or read.

In parallel, each processor builds its own ISAT table. If **Verbosity** is enabled in parallel, each compute node writes out the **Verbosity** file(s) with the node ID number appended to the file name.

19.1.3. Using ISAT Efficiently

Efficient use of ISAT requires thoughtful control. What follows are some detailed recommendations concerning the achievement of this goal.

Important

- The numerical error in the ISAT table is controlled by the **ISAT Error Tolerance**, which has a default value of 0.001. This value is relatively large, which allows faster convergence times for steady-state simulations. However, once the solution has converged, it is important to reduce this **ISAT Error Tolerance** and re-converge. This process should be repeated until the species that you are interested in modeling are unchanged. Unsteady simulations should be run with a sufficiently small **ISAT Error Tolerance** so that the species of interest are unaffected by this parameter. Note that as the error tolerance is decreased, the memory and time requirements to build the ISAT table will increase substantially. There is a large performance penalty in specifying an error tolerance smaller than is needed to achieve acceptable accuracy, and the error tolerance should be decreased gradually and judiciously.
- As described in Mesh Partitioning and Load Balancing (p. 2030), you can select portions of the simulation to consider when performing dynamic load balancing in multiprocessor simulations. If the **ISAT** option is selected in the **Weighting** tab of the Partitioning and Load Balancing Dialog Box (p. 2767), the time required for solving chemistry will be factored in when assigning the computational cells to each available processor. It is recommended that you select this option when solving stiff chemistry, particularly when the Dynamic Mechanism Reduction method is enabled; thus, more resources can be allocated for the cells with the larger mechanisms.

During the initial iterations, before a steady-state solution is attained, transient composition states occur that are not present in the steady-state solution. For example, you might patch a high temperature region in a cold fuel-air mixing zone to ignite the flame, whereas the converged solution never has hot reactants without products. Since all states that are realized in the simulation are tabulated in ISAT, these initial mappings are wasteful of memory, and can degrade ISAT performance. If the table fills the allocated memory and contains entries from an initial transient that are no longer accessed, it may be beneficial to purge the ISAT table. This is achieved by either clearing it in the **Integration Parameters** dialog box, or saving your case and data files, exiting ANSYS Fluent, then restarting ANSYS Fluent and reading in the case and data. This can also be done with the TUI command `define/models/species/clear-isat-table`.

From experience, ISAT performs very well on premixed turbulent flames, where the range of composition states are smaller than in non-premixed flames. ISAT performance degrades in flames with large residence times, where more work is required in the ODE integrator.

19.1.4. Reading and Writing ISAT Tables

ANSYS Fluent can write and subsequently read ISAT tables. However, it is in general not recommended to write and read ISAT tables for the following reason: ISAT tabulates chemical states that are specific

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

to a single simulation. The ISAT tabulated composition states are determined by the geometry, boundary conditions, physics models such as turbulence and radiation, thermodynamic and transport properties, as well the chemical mechanism. If any of these parameters change, the realized composition space changes, and significant parts of the existing ISAT table are no longer accessed. These un-accessed ISAT table entries slow interpolation and decrease the number of useful accessed table entries that can be added. Since the time consumed building the ISAT table is typically much smaller than simulation run times, it is advised to rebuild the table when restarting.

When ANSYS Fluent is run in parallel, each partition builds its own ISAT table and does not exchange information with ISAT tables on other compute nodes. You can save the ISAT tables on all compute nodes:

File → Write → ISAT Table...

Each compute node writes out its ISAT table to the specified file name, with the node ID number appended to the file name. For example, a specified file name of `my_name` on a two compute node run will write two files called `my_name-0.isat` and `my_name-1.isat`.

Subsequent runs can start from existing ISAT tables by reading them into memory.

File → Read → ISAT Table...

Files can be read in two ways:

- Parallel nodes can read in corresponding ISAT tables saved from a previous parallel simulation. The appended node ID should not be removed from the input file name.

  Note

  The ability to use ISAT tables generated from a parallel simulation with a different number of parallel nodes is not supported.

- All nodes can read one unique ISAT table. You might use this approach if you have a large table from a serial simulation. ANSYS Fluent first checks to see if the exact file name that you specified exists, and if it does, all nodes will read this one file.

19.2. Using Dynamic Mechanism Reduction

Computational time increases with the size of the chemical kinetics mechanism used in the simulation. Dynamic Mechanism Reduction (DMR) decreases the mechanism size at each cell (or particle) to include only those species and reactions necessary for accurate modeling (within defined tolerances) of the chemical kinetics at the local conditions. This is performed at every flow iteration (for steady simulations) or time step (for transient simulations). Because the mechanism is only required to be accurate at the local conditions, smaller reduced mechanisms can be used with less accuracy loss than when using skeletal mechanism reduction, where a single reduced mechanism is used throughout the simulation. The speed-up from this approach is problem-dependent, but on average is about a factor of two or more compared to a case with no chemistry acceleration.

For an even greater increase in performance, you can use Dynamic Mechanism Reduction with any combination of the other chemistry acceleration options available in ANSYS Fluent. In most cases, applying ISAT and DMR together should give the best performance. Note, however, that ISAT performance

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

degrades when accuracy requirements are very strict or when conditions are not revisited often in a simulation. An example is unsteady ignition simulations, where ISAT table entries may not be re-used. However, the important species and reactions can vary widely during the simulation; for example, many of the low temperature ignition reactions are only required for the small spark zone and short spark duration. In such situations, you may achieve better performance using Dynamic Mechanism Reduction along with direct integration method rather than ISAT.

Combining multiple chemistry acceleration methods, however, compounds accuracy loss, since each of these methods accelerates the simulation by sacrificing some accuracy. In general, greater speed-up is achievable when the full mechanism is larger, because there is potential for a greater degree of reduction.

Important

When using Dynamic Mechanism Reduction along with ISAT, setting the error tolerance for either approach above its default value degrades accuracy faster than when either method is used alone.

Dynamic Mechanism Reduction is performed using the Directed Relation Graph (DRG) algorithm. For details of the DMR algorithm, refer to the discussion in Dynamic Mechanism Reduction in the *Fluent Theory Guide*.

You can enable Dynamic Mechanism Reduction through either the graphical user interface (GUI) or the text user interface (TUI) as follows.

- In the GUI, enable the **Dynamic Mechanism Reduction** option in the **Integration Parameters** dialog box.
- In the TUI, enter the following text command in the console:

  ```
  define/models/species/integration-parameters
  ```

  When prompted with `Enable Dynamic Mechanism Reduction?`, answer with `yes`.

  Note that it is not necessary to enable Chemistry Acceleration Expert mode in order to use Dynamic Mechanism Reduction.

For additional information, see the following sections:
19.2.1. Mechanism Reduction Parameters
19.2.2. Monitoring and Postprocessing Dynamic Mechanism Reduction
19.2.3. Using Dynamic Mechanism Reduction Effectively

19.2.1. Mechanism Reduction Parameters

In ANSYS Fluent, Dynamic Mechanism Reduction is controlled by the error tolerance and the target species list, as described below.

Error Tolerance

(ε in Equation 12.4 in the *Fluent Theory Guide*).The default value for error tolerance ε is `0.01`, which should work well for most simulations, balancing speed and accuracy. However, a larger tolerance may be sufficient for some simulations (for example, steady-state simulations with moderate accuracy requirements), while a smaller tolerance may be required in other cases with stricter accuracy requirements (for example, modeling auto-ignition delay times for a highly complex fuel). In general, a larger tolerance yields faster, but less accurate simulations.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The error tolerance can be adjusted through the GUI or TUI as follows:

- In the GUI, set **Error Tolerance** to the desired value in the **Mechanism Reduction Parameters** group box in the **Integration Parameters** dialog box.
- In the TUI, enter the following text command: `define/models/species/integration-parameters`. When prompted with `Mechanism Reduction Error Tolerance`, enter the desired tolerance (a positive, nonzero value) or press **Enter** to retain the default value.

Note

- In the TUI, you can also enable Chemistry Acceleration Expert mode, other chemistry acceleration methods, and specify ODE integrator parameters (see Using ISAT (p. 1128) for details about these parameters).
- Chemistry Acceleration Expert mode should only be enabled if you want to modify the default target species list.

Target Species List

As described in the Dynamic Mechanism Reduction in the *Fluent Theory Guide*, the target species are those species that you want to predict most accurately. The default number of target species N_{target} is 3. The first default target species is hydrogen radical. DRG will add two other species with the largest mass fractions to complete the target species list at each flow time step or iteration. DRG will then identify all other species that must also be included in the mechanism in order to accurately model these targets.

Although you should rarely need to alter the default target species list, ANSYS Fluent provides an option to specify target species of your choice. The designated target species can be specified only in the TUI when Chemistry Acceleration Expert mode is enabled. Enter the text command `define/models/species/integration-parameters` and then answer `yes` when prompted with `Enable Chemistry Acceleration expert?`.

ANSYS Fluent provides the ability to explicitly specify target species. From your mixture material list, you can select any number N_{user} of target species to be included in the kinetics mechanism. In case no targets were selected (N_{user}=0) or the number of user-selected targets N_{user} is less than the minimum number of target species N_{target} (N_{user}<N_{targets}), the DRG algorithm will add the (N_{targets}−N_{user}) species with the largest mass fractions to the target species list.

As discussed in the Dynamic Mechanism Reduction in the *Fluent Theory Guide*, there is an option to remove a species from the target list whenever its mass fraction is below a specified threshold. This option is disabled by default in ANSYS Fluent (that is, the default value for minimum mass fraction is 0).

When specifying Target Species List, you will receive the following prompts in the console:

1. `Minimum number of target species [3]`

   Enter the desired number of targets.

2. `Minimum mass fraction of target species [0]`

   Enter minimum allowable target mass fraction.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The current target species list will be displayed in the console window (for example, the console output for the default target species consisting of hydrogen radical will be `Current target species list = (h)`).

3. `Enter target species list...`

   `Species name [""]`

   Enter target species name, for example, **"ch4"**.

   Important

   You *must* enter the complete name of the species as it appears in the **Chemical Formula** field in the **Create/Edit Materials** dialog box within quotes (" ").

   After you enter the first target species, you will be prompted to specify the second one, and so on, until you press **Enter** to complete the setup.

   Note, that if you have entered **0** for the minimum number of target species N_{target} and have not provided any target species (that is, the number of user-selected targets, N_{user}, is zero), ANSYS Fluent will automatically select the three species with the largest mass fraction and use them as the target species in the DRG algorithm.

19.2.2. Monitoring and Postprocessing Dynamic Mechanism Reduction

When Chemistry Acceleration expert is enabled in the TUI, two additional field variables are available to be monitored or examined in postprocessing:

- **DRG Reduced Number of Species** in the **Species...** category
- **DRG Reduced Number of Reactions** in the **Reactions...** category

These field variables quantify the size of the reduced mechanism at each cell or particle in the domain (the number of retained species and reactions, respectively).

When you use DMR in combination with ISAT, ANSYS Fluent will report values of zero for **DRG Reduced Number of Species** and **DRG Reduced Number of Reactions** for those cells where the ANSYS solver computed the solution using table lookup instead of direct integration in the ISAT algorithm. A cell value of zero **DRG Reduced Number of Species** does not imply that the DMR algorithm eliminated all species from the mechanism; rather, it indicates that the ANSYS solver performed ISAT table lookup to obtain the chemistry solution at the cell.

If you want to view the cells for which ISAT table lookup was performed in the last iteration or time step, display the **DRG Reduced Number of Species** plot clipped to a range of 0 to 0. (Make sure that the **Node Value** option is deselected in the postprocessing dialog boxes.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you want to view the cells for which DMR was performed in the last iteration or time step, display the **DRG Reduced Number of Species** plot clipped to a range of 1 to a global maximum value (as it appears in the **Max** field when **Global Range** is selected in the postprocessing dialog boxes).

Note

The DMR postprocessing field variables are not stored in the data file and, hence, will not be available for postprocessing in the next session. If you want to postprocess these variables in your next session, read the data file and perform a single iteration or time step (for steady-state or transient simulations, respectively) in order for the DMR data to be calculated and available for viewing.

19.2.3. Using Dynamic Mechanism Reduction Effectively

As described in Dynamic Mechanism Reduction in the *Fluent Theory Guide*, mechanism reduction is performed at each cell or particle, once per time step (for transient simulations) or per flow iteration (for steady-state simulations). It is assumed that the reduced mechanism created for the starting conditions will remain valid for the entire transport time step over which the chemistry ODE is integrated. If the chemistry integration time interval is very large, the chemical state can change significantly, degrading the accuracy of the reduced mechanism. In some cases (particularly transient simulations such as ignition) it may be necessary to enforce smaller flow time steps in order to effectively use Dynamic Mechanism Reduction. In this way the reduced mechanisms are updated more frequently to match the changing chemical states.

Note that Dynamic Mechanism Reduction works best when there are significant regions in a computational domain and/or times during a simulation with relatively low chemical activity (for example, low temperature, low mixing of fuel/oxidizer, low concentrations of reactive species, and so on). Consider a simple opposed flow diffusion flame problem with pure fuel as one stream and pure oxygen as the other stream. Large mechanisms (which, depending on the error tolerance, may be close to the full mechanism size) will likely be used in the mixing region where the flame is located, while smaller mechanisms would be used elsewhere in the domain. You should not expect much speed-up if a very large fraction of your grid cells (for example, 90%) are in the flame region.

In steady-state simulations, you can improve the solution time by first running to convergence with a larger error tolerance ε, and then restarting the simulation repeatedly, gradually decreasing the error tolerance ε until you reach the desired accuracy level. For example, if you first converge the solution with the error tolerance ε=0.1, then iterate the solution further to convergence with the lower error tolerance, ε=0.05, and then with the desired error tolerance ε=0.01, you will generally be able to achieve faster convergence than iterating from initial conditions with the error tolerance ε=0.01.

Important

As described in Mesh Partitioning and Load Balancing (p. 2030), you can select portions of the simulation to consider when performing dynamic load balancing in multiprocessor simulations. When using the Dynamic Mechanism Reduction along with ISAT, it is recommended that you select the **ISAT** option in the **Weighting** tab of the Partitioning and Load Balancing Dialog Box (p. 2767) when solving stiff chemistry. If **ISAT** is selected, the time required for solving chemistry will be factored in when assigning the computational cells to each available processor; thus, more resources can be allocated for the cells with the larger mechanisms.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

19.3. Using Chemistry Agglomeration

The Chemistry Agglomeration method reduces the number of calls to the computationally expensive ODE integrator by clustering cells with similar compositions. The size of these clusters is determined by the agglomeration parameters **Error Tolerance** (ε_{tol}) and **Temperature Bin** (Δ_T) in the **Integration Parameters** dialog box. Larger values of ε_{tol} and Δ_T result in a larger number of agglomerated cells, fewer calls to the reaction integrator, increased run-time speed, but greater error.

You can also set the chemistry agglomeration parameters using the text user interface (TUI) command `define/models/species/integration-parameters`. Once you enable agglomeration chemistry, ANSYS Fluent prompts you for `Agglomerate Chemistry Error Tolerance` and `Agglomerate Chemistry Temperature Bin`. Enter the desired values or press **Enter** to retain the default values of `0.02` for `Agglomerate Chemistry Error Tolerance` and `10` for `Agglomerate Chemistry Temperature Bin`.

More information can be found in Chemistry Agglomeration in the *Fluent Theory Guide*.

Important

The solution accuracy can be improved by decreasing the tolerances. Generally, it should not be necessary to reduce the tolerances below 0.01 and 10 K.

19.4. Dimension Reduction

Detailed kinetic mechanisms typically contain a multitude of intermediate species that far exceed the number of major fuel, oxidizer, and product species. Chemical mechanism **Dimension Reduction** reduces the number of intermediate species transport equations (called representative species) that are solved, and reconstructs the 'unrepresented' species using chemical equilibrium assumptions.

Important

Since ANSYS Fluent is limited to a maximum of 700 transported species, the main use of **Dimension Reduction** is to enable simulation with chemical mechanisms containing more than 700 species.

Follow these steps to use **Dimension Reduction**:

- Import your CHEMKIN mechanism. Note that you can import CHEMKIN mechanisms that contain more than 700 species.
- Click the **Integration Parameters** button in the **Species Model** dialog box, and enable **Dimension Reduction**.
- Set the **Number of Represented Species**. This must be greater than 10 and less than the number of species in the full mechanism. The **Number of Represented Species** must also be less than 700 minus the number of unrepresented elements (the number of chemical elements in the unrepresented species). A larger **Number of Represented Species** will increase accuracy, but also increase computational expense. The default of 12 has been chosen to provide a good compromise between accuracy and speed.
- Select the **Full Mechanism Material Name**, which is typically the name of the CHEMKIN mechanism that you imported. ANSYS Fluent can store several imported CHEMKIN mechanisms in different mixture materials

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

whose mechanisms you are not using. However, since mechanisms are typically large, you should delete unused mixtures to reduce memory requirements.

- Set the boundary and initial fuel and oxidizer, as well as product species, as represented species. These are set in the **Fuel/Oxidizer Species** list. Note that you can force other species to be represented by selecting them here. Species of interest, especially species that are not near chemical equilibrium, such as pollutants, and their associated intermediate species in the mechanism should also be included. Intermediate species that occur in large mass fractions relative to the fuel and oxidizer species should be included, as well as species important in the chemical pathway. For example, for methane combustion in air, CH_3 should be included as a represented species since CH_4 pyrolizes to CH_3 first.

- Click **Calculate Reduced Dimension Mixture**. This will create a new mixture material called `reduced-dimension-mixture`, which contains the represented species as well as proxy 'species' for the unrepresented elements. These unrepresented elements have "u" prepended to the element name. You should never rename the reduced-dimension-mixture mixture material, or select another mixture as the active material, while **Dimension Reduction** is enabled.

- Continue with the setup, solution, and postprocessing as for other detailed chemistry cases. Note that the boundary and initial mass fraction of unrepresented elements should always be zero. The entries for the unrepresented element mass fractions are disabled in the ANSYS Fluent GUI.

For postprocessing, the unrepresented elements are available in the **Species** list as the element name with "u" prepended. All species in the full mechanism, consisting of both represented and unrepresented species, are available in the **Full Mechanism Species...** category of the postprocessing dialog boxes.

After you obtain a preliminary solution with **Dimension Reduction**, it is recommended that you check the magnitude of all unrepresented species, which are available in the **Full Mechanism Species...** option in the **Contours** dialog box. If the mass fraction of any unrepresented species is larger than other represented species, you should repeat the simulation with this species included in the represented species list. In turn, the mass fraction of all unrepresented elements should decrease.

Note

Note that **Dimension Reduction** is only available with ISAT. **Dimension Reduction** is initially comparable in speed to a simulation with the full mechanism, but iterations become significantly faster at later times when the ISAT table is populated.

For information about the theory of this option, see Chemical Mechanism Dimension Reduction in the *Fluent Theory Guide*.

19.5. Using Dynamic Cell Clustering

The Dynamic Cell Clustering (DCC) method is available with the transient ANSYS CHEMKIN-CFD solver. DCC groups computational cells with similar temperatures, pressures, and initial species mass fractions into clusters.

Two parameters are available for controlling cell clustering:

- **Max. Temperature Dispersion** (default =10 K)
- **Max. Equiv. Ratio Dispersion** (default =0.05)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The default values for these parameters have been shown to provide accurate results for a wide range of combustion cases. Reducing these criteria will generally result in more clusters being created, and therefore more computational time.

The method is enabled by default and is strongly recommended for improving computational efficiency. For background information about DCC, refer to Dynamic Cell Clustering with ANSYS CHEMKIN-CFD Solver in the *Fluent Theory Guide*.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 20: Modeling Engine Ignition

This chapter discusses how to use the engine ignition models available in ANSYS Fluent in the following sections. For information about the theory behind these ignition models, see Engine Ignition in the Theory Guide.

20.1. Spark Model
20.2. Autoignition Models
20.3. Crevice Model

20.1. Spark Model

The spark model in ANSYS Fluent will be described in the context of the premixed turbulent combustion model. For information regarding the theory of this model, see Spark Model in the Theory Guide. Information regarding the use of this model is detailed in the following sections:

20.1.1. Using the Spark Model
20.1.2. Using the ECFM Spark Model

20.1.1. Using the Spark Model

You can model a single spark or multiple sparks. To use the spark model, perform the following steps:

1. Select **Transient** from the **Time** list in the **General** task page (or from the **General** → **Analysis Type** in the tree).

2. In the **Species Model** dialog box, select one of the following reaction models:

   Setup → **Models** → **Species** **Edit...**

   - Under **Model**, select **Species Transport**, and under **Reactions**, enable **Volumetric**.
   - Under **Model**, select **Premixed Model** with the **C Equation** or **G Equation** model enabled. See Setting Up the C-Equation and G-Equation Models (p. 1092) for more information.

   The **Spark Ignition** model will now appear in the **Models** task page and under the **Models** tree item.

   Important

   When you read in a R14.5 case file, ANSYS Fluent will revert to the R14.5 spark model by default. To switch to the current spark model, you can either

   - open the **Spark Ignition** dialog box, or
   - use the TUI `define/models/species/spark-model` command.

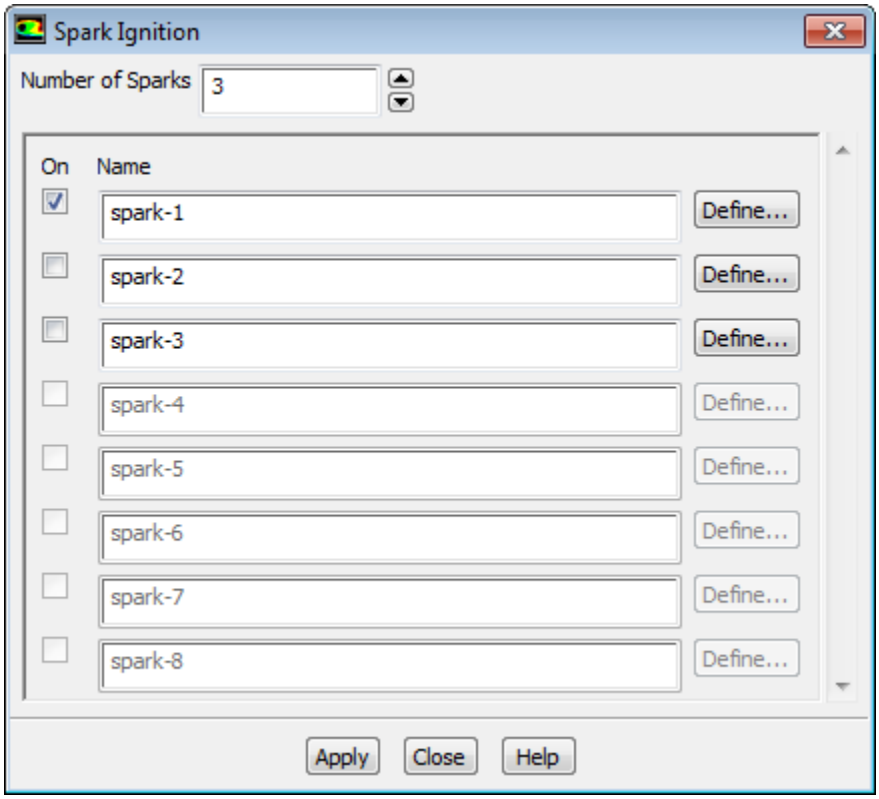
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also use the TUI `define/models/species/spark-model` command to revert back to the R14.5 spark model.

3. Define sparks for your case using the **Spark Ignition** dialog box (see Figure 20.1: The Spark Ignition Dialog Box (p. 1140)).

   Setup → Models → Species → Spark Ignition Edit...

   Figure 20.1: The Spark Ignition Dialog Box



4. Specify the **Number of Sparks** you would like to include in your simulation. You can define up to 16 sparks.

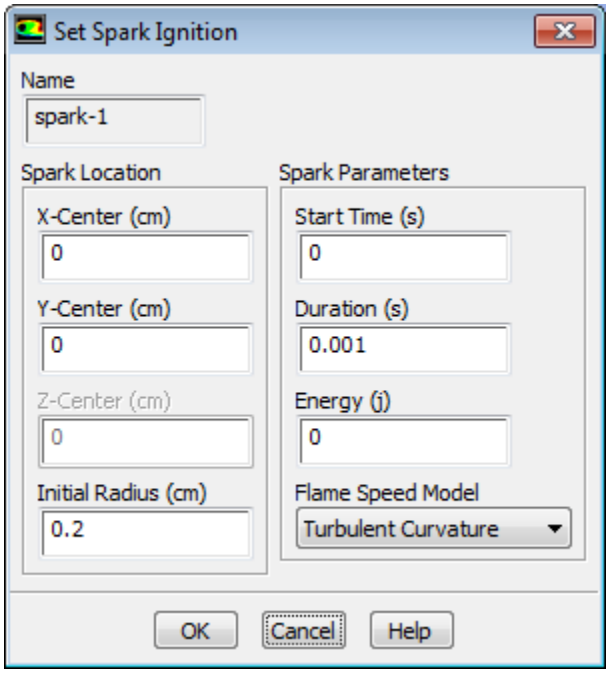
5. While you can define several sparks, you can choose which ones to turn on using the **On** option.

6. Enter the **Name** of the spark, or simply keep the default name.

7. Click the **Define...** button to open the **Set Spark Ignition** dialog box (Figure 20.2: The Set Spark Ignition Dialog Box (p. 1141)), where you will set the parameters of the selected spark ignition models.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 20.2: The Set Spark Ignition Dialog Box



8. Set the spark model parameters.

   - Set X, Y, and Z coordinates of the spark center and the **Initial Radius** of the spark kernel in the **Spark Location** group box.
   - Enter **Start Time** and **Duration** in the **Spark Parameters** group box.

     Important

     When the in-cylinder model is turned on, the **Start Time** is entered in crank angle degrees instead of seconds, while the spark **Duration** is still in seconds. The value you enter for **Start Time** is for one complete engine cycle. In subsequent cycles, the spark is enabled according to Equation 10.30 (p. 716), where θ_{event} corresponds to the spark start crank angle.

   - (optional) Set **Energy** to a positive value if you want to model the higher temperature levels within the spark kernel. The energy input will be evenly distributed across the specified spark duration. This energy is not required to initiate combustion. The spark model will control the spark kernel growth and combustion progress. For this reason the default energy input is zero. If you specify extra energy here, it will raise the kernel temperature beyond that given by the combustion process.
   - Select the **Flame Speed Model** from the drop-down list. For details of these models see Spark Model Theory in the *Fluent Theory Guide*.

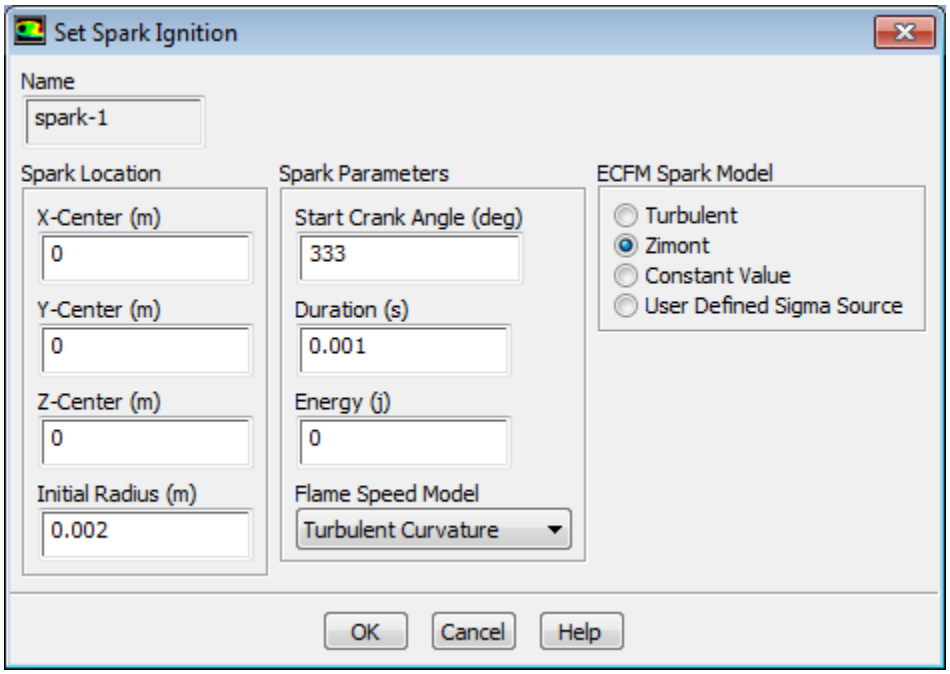
20.1.2. Using the ECFM Spark Model

When the **Extended Coherent Flamelet Model** (see Setting Up the Extended Coherent Flame Model (p. 1094)) is selected in the **Species Model** dialog box, the **Set Spark Ignition** dialog box will exhibit

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

additional **ECFM Spark Model** options, as shown in Figure 20.3: The Set Spark Ignition Dialog Box Displaying the ECFM Spark Model Options (p. 1142).

Figure 20.3: The Set Spark Ignition Dialog Box Displaying the ECFM Spark Model Options



In addition to specifying the variables mentioned in Using the Spark Model (p. 1139), you can select from one of the following ECFM Spark models:

- **Turbulent**
- **Zimont** (default)
- **Constant Value**
- **User Defined Sigma Source**

This controls the way in which the value of the flame surface density is calculated. More details are given in ECFM Spark Model Variants in the *Fluent Theory Guide*.

If you choose **Constant Value** you will be required to specify the value for flame surface density. If you choose **User Defined Sigma Source** you will need to choose the hooked user-defined function to apply a custom source term to the ECFM equation within the volume of the spark ignition kernel. For more detail about this user-defined function, refer to Hooking `DEFINE_ECFM_SPARK_SOURCE` UDFs in the *Fluent Customization Manual*.

20.2. Autoignition Models

Autoignition phenomena in engines are due to the effects of chemical kinetics of the reacting flow inside the cylinder. There are two types of autoignition models considered in ANSYS Fluent:

- knock model in spark-ignited (SI) engines
- ignition delay model in diesel engines

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For information regarding the theory behind autoignition models, see Autoignition Models in the Theory Guide. Using the Autoignition Models (p. 1143) describes how to use the autoignition models in ANSYS Fluent.

20.2.1. Using the Autoignition Models

To use the autoignition model, perform the following steps:

1. Select **Transient** from the **Time** list in the **General** task page (or from the **General** → **Analysis Type** in the tree).

2. Select an appropriate reaction model in the **Species Model** dialog box.

   Setup → **Models** → **Species** **Edit...**

3. The models in the **Species Model** dialog box that are compatible with the autoignition model are **Species Transport**, **Premixed Combustion**, and **Partially Premixed Combustion**.

   The **Autoignition** model will now appear in the **Models** task page and under the **Models** tree item.

   Important

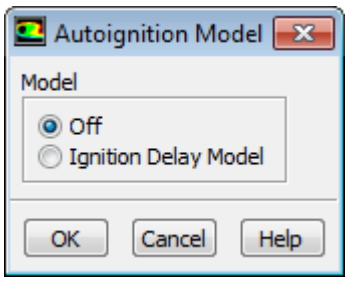
   - If you select **Species Transport**, you must also enable the **Volumetric** option in the **Reactions** group box.
   - The **Premixed Combustion** and **Partially Premixed Combustion** models are only available for turbulent flows using the pressure-based solver.

4. Select the Autoignition model.

   Setup → **Models** → **Species** → **Autoignition** **Edit...**

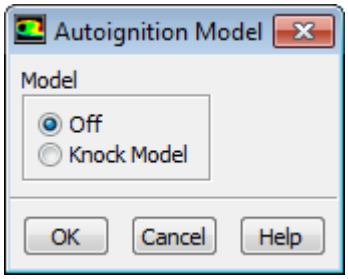
   - If **Species Transport** is selected in the **Species Model** dialog box, you can only select the **Ignition Delay Model**.

     Figure 20.4: The Ignition Delay Model in the Autoignition Model Dialog Box

   - If **Premixed Combustion** is selected in the **Species Model** dialog box, you can only select the **Knock Model**.

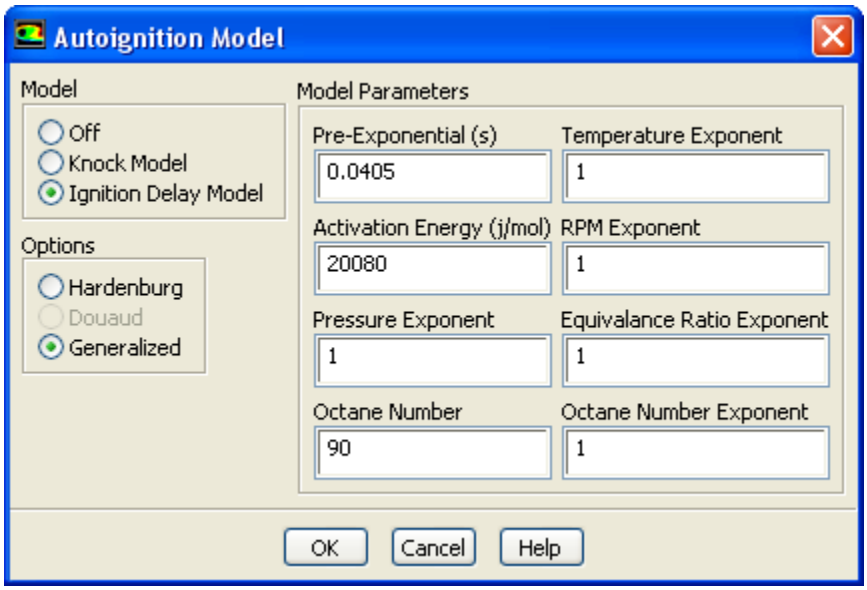
Figure 20.5: The Knock Model in the Autoignition Model Dialog Box



- If **Partially Premixed Combustion** is selected in the **Species Model** dialog box, you can select either the **Knock Model** or the **Ignition Delay Model**.

5. When the **Ignition Delay Model** is enabled, the dialog box expands to include the modeling parameters for this model (Figure 20.6: The Ignition Delay Model for the Partially Premixed Combustion Model (p. 1144)). The two correlation options that exist with this model are **Hardenburg** and **Generalized**. Depending on which correlation option is selected, the appropriate modeling parameters will appear in the dialog box.

Figure 20.6: The Ignition Delay Model for the Partially Premixed Combustion Model



- The **Hardenburg** option is typically used for heavy duty diesel engines. With this option, the following parameters are available:
  - **Pre-Exponential**
  - **Pressure Exponent**
  - **Activation Energy**
  - **Cetane Number**

  For the **Species Transport** model, **Fuel Species** is selected from the drop-down list.

- The **Generalized** option is described by Equation 13.10 in the Theory Guide. With this option, the following parameters are available:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

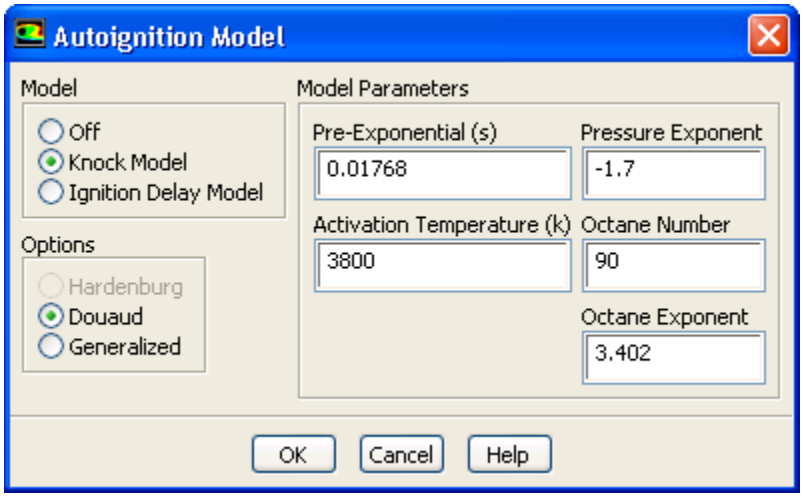
- **Pre-Exponential**
- **Temperature Exponent**
- **Activation Energy**
- **RPM Exponent**
- **Pressure Exponent**
- **Equivalence Ratio Exponent**
- **Octane Number**
- **Octane Number Exponent**

For the **Species Transport** model, **Fuel Species** is selected from the drop-down list.

Default values of these parameters can be found in Default Values of the Variables in the Hardenburg Correlation in the *Fluent Theory Guide*.

6. When the **Knock Model** is enabled, the dialog box expands to include modeling parameters for this model (Figure 20.7: The Knock Model with the Partially Premixed Combustion Model Enabled (p. 1145)). The two correlation options that exist with this model are **Douaud** and **Generalized**. Depending on which correlation option is selected, the appropriate modeling parameters will appear in the dialog box.

Figure 20.7: The Knock Model with the Partially Premixed Combustion Model Enabled



- The **Douaud** option is used for knock in SI engines. The modeling parameters that are specified in the **Autoignition Model** dialog box for this option are the **Pre-Exponential**, **Pressure Exponent**, **Activation Temperature**, **Octane Number**, and **Octane Exponent** (Equation 13.9 in the Theory Guide).
- The **Generalized** option (Equation 13.10 in the Theory Guide) in the knock model requires the same parameters as in the ignition delay model.

20.3. Crevice Model

For information regarding the theory behind the crevice model, see Crevice Model in the Theory Guide. Using the crevice models in ANSYS Fluent are described in the following sections:

20.3.1. Using the Crevice Model

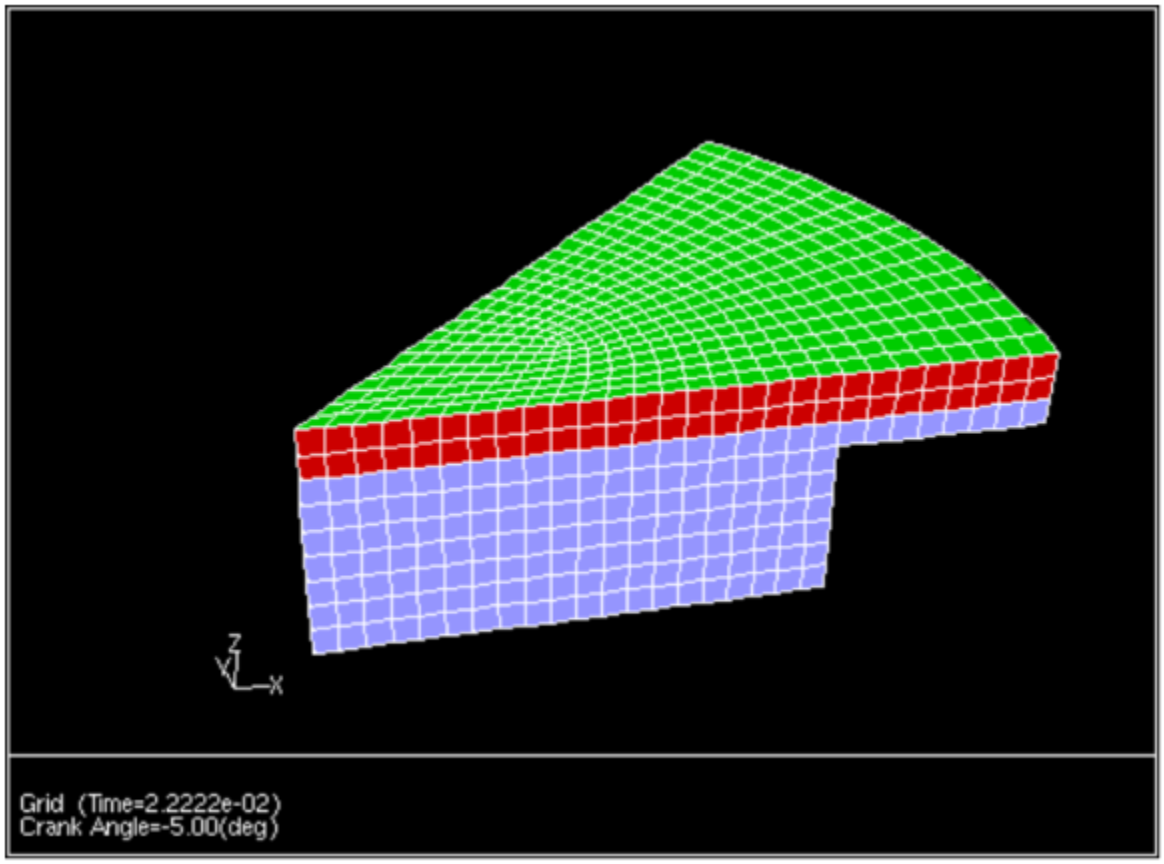
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20.3.2. Crevice Model Solution Details
20.3.3. Postprocessing for the Crevice Model

20.3.1. Using the Crevice Model

An optical experimental engine [25] (p. 2888) is used below to show a working example of how to use the crevice model as it is implemented in ANSYS Fluent. The mesh at ten crank angle degrees before top center is shown in Figure 20.8: Experimental Engine Mesh (p. 1146).

Figure 20.8: Experimental Engine Mesh



The following example shows the necessary steps to enable the crevice model for a typical in-cylinder flow.

1. From the > prompt, enter the `define/models` menu by using the following text command:

   `define` → `models`

2. Enable the crevice model, as follows:

```
/define/models   crevice-model?

Enable crevice model? [no] yes
/define/models
acoustics/                multiphase/                    species/
addon-module              noniterative-time-advance?     steady?
axisymmetric?             nox?                           unsteady-1st-order?
crevice-model-controls/   radiation/                     unsteady-2nd-order?
crevice-model?            solidification-melting?        viscous/
dpm/                      solver/
energy?                   soot?
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
   frozen-flux?              sox?
```

3. Enter the ring pack geometry:

```
   /define/models   crevice-model-controls

   Cylinder bore (m) [0.1] 0.1397
   Piston to bore clearance (m) [3.0e-5] 5.08e-05
   Piston crevice temperature (K) [400] 433
   Piston sector angle (deg) [360] 45
   Ring discharge coefficient [0.8] 0.7
   Pressure in crankcase (exit pressure) (Pa) [101325]
   Write  out  crevice  data  to  a  file?  [no]   yes

   output file name ["crev.out"]

 Available  wall  threads  are:  (wall.1  wall  wall-8)
 Leaking  wall []   wall.1

Shared boundary [] wall-8
   Selected  boundary  threads  :  (wall.1  wall-8)
  Use  these  zones?  [yes]   yes

Solve crevice model ? [no] yes
Number of rings [3]
           Width  of  ring  number  0  is:  [0.00375]
       Thickness  of  ring  number  0  is:  [0.0015]
         Spacing  of  ring  number  0  is:  [0.008]
   Land  Length  for  ring  number  0  is:  [0.00391]
        Top  Gap  of  ring  number  0  is:  [6e-05]
     Middle  Gap  of  ring  number  0  is:  [4e-05]
     Bottom  Gap  of  ring  number  0  is:  [6e-05]

           Width  of  ring  number  1  is:  [0.00375]
       Thickness  of  ring  number  1  is:  [0.0015]
         Spacing  of  ring  number  1  is:  [0.008]
   Land  Length  for  ring  number  1  is:  [0.00391]
        Top  Gap  of  ring  number  1  is:  [6e-05]
     Middle  Gap  of  ring  number  1  is:  [4e-05]
     Bottom  Gap  of  ring  number  1  is:  [6e-05]

           Width  of  ring  number  2  is:  [0.00375]
       Thickness  of  ring  number  2  is:  [0.0015]
         Spacing  of  ring  number  2  is:  [0.00391]
   Land  Length  for  ring  number  2  is:  [0.00391]
        Top  Gap  of  ring  number  2  is:  [6e-05]
     Middle  Gap  of  ring  number  2  is:  [4e-05]
     Bottom  Gap  of  ring  number  2  is:  [6e-05]


   Initial  conditions  in  ring  pack
   Pressure  1  is:  [4600623.5]
   Pressure  2  is:  [4173522.5]
   Pressure  3  is:  [3689110.5]
   Pressure  4  is:  [3130620]
   Pressure  5  is:  [2214841.8]
```

A fast way to set up multiple rings in the ring pack is to specify only one ring and enter the geometry. Once the ring geometry is entered, invoke the `crevice-model- controls` menu a second time and specify the number of rings desired. When the number of rings changes, the geometry from the first ring is copied to all subsequent rings. Default values can be taken for the rest of the way through the menu structure.

A summary of the crevice model is printed out by entering the `(crevice-summary)` command at the command prompt:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
>(crevice-summary)
                    crevice/n-rings  :  3
                  crevice/ring-width  :  (0.00375  0.00375  0.00375)
              crevice/ring-thickness  :  (0.0015  0.0015  0.0015)
                   crevice/ring-mass  :  (0.00375  0.00375  0.00375)
                crevice/ring-spacing  :  (0.008  0.008  0.00391)
                 crevice/land-length  :  (0.00391  0.00391  0.00391)
                crevice/top-ring-gap  :  (6e-05  6e-05  6e-05)
                crevice/mid-ring-gap  :  (4e-05  4e-05  4e-05)
                crevice/bot-ring-gap  :  (6e-05  6e-05  6e-05)
           crevice/piston-temperature  :  433
                 crevice/sector-angle  :  45
                   crevice/mid-gap-cd  :  0.7
                crevice/exit-pressure  :  101325
                      crevice/threads  :  (5  6)
           names  of  crevice/threads  :  (wall.1  wall-8)
                crevice/unit-roundoff  :  5.9604645e-08
       crevice/piston-bore-clearance  :  5.08e-05
                      crevice/write?  :  #t
                crevice/output-file  :  crev.out
                      crevice/solve?  :  #t
                    crevice/enabled?  :  #t
                    crevice/pressures  :  (4600623.5  4173522.5  3689110.5  3130620  2214841)
```

20.3.2. Crevice Model Solution Details

The under-relaxation factor for the crevice model source terms can be found in the **Solution Controls** task page. The default value for **Crevice Model Sources** is 0.8, which has been found to work well for motored engine simulations. Once the crevice model is enabled, the solution proceeds normally.

Solution → **Controls**

Solution → **Initialization**

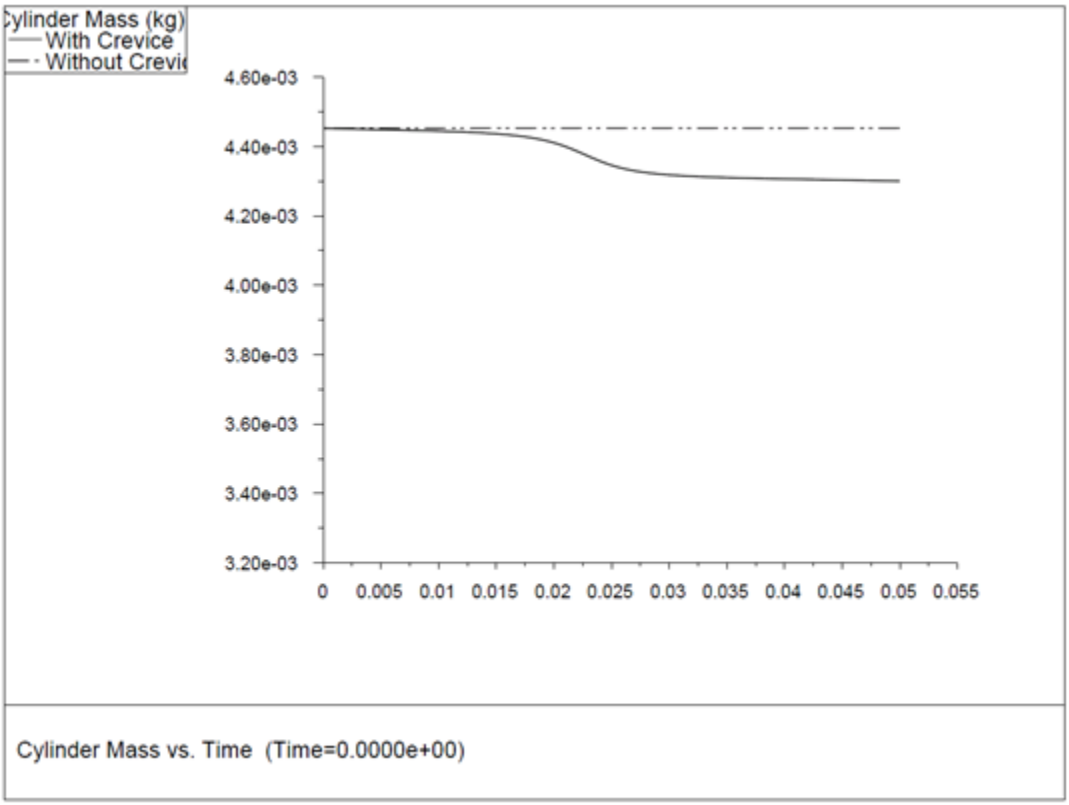
Solution → **Run Calculation**

20.3.3. Postprocessing for the Crevice Model

A plot of cylinder mass with and without the crevice model during the motored engine simulation is shown in Figure 20.9: Cylinder Mass vs. Crank Angle (p. 1149). The rate of mass loss from the crevice is proportional to the pressure difference between the cylinder and the crankcase pressure defined in the text interface.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

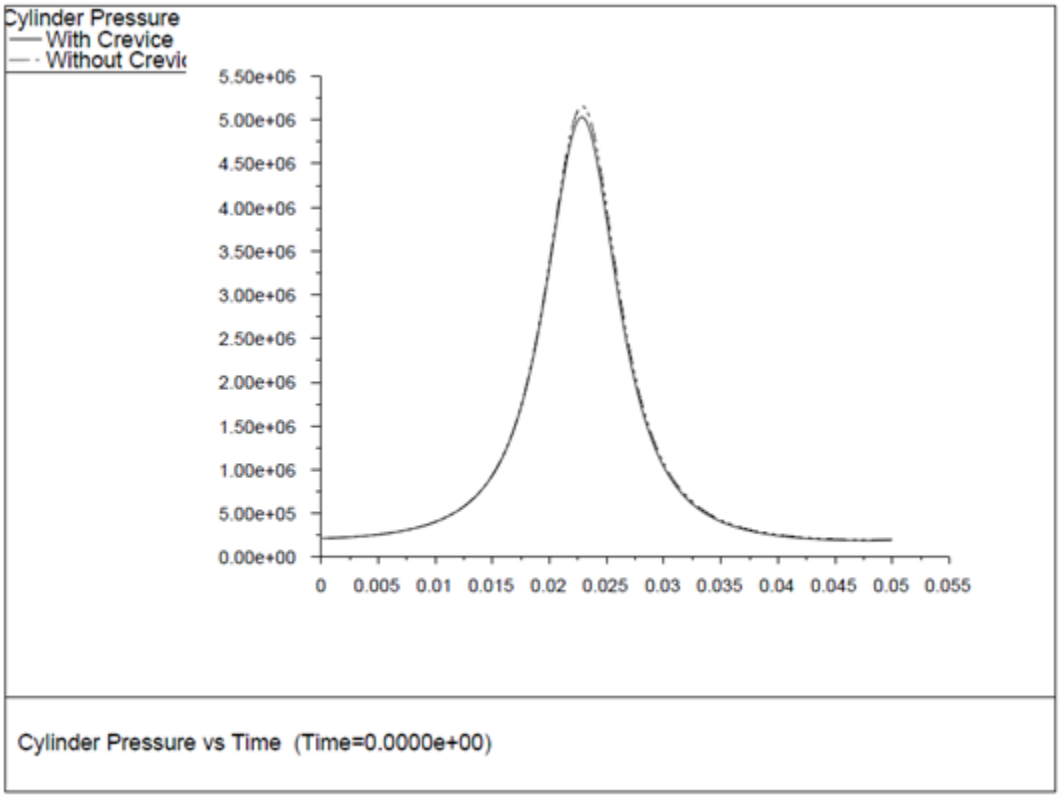
Figure 20.9: Cylinder Mass vs. Crank Angle



A plot of cylinder pressure with and without the crevice model for the same engine simulation is shown in Figure 20.10: Cylinder Pressure vs. Crank Angle (p. 1150). The effect of the mass loss from the crevice is to lower the peak pressure in proportion to the total mass loss from the cylinder.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 20.10: Cylinder Pressure vs. Crank Angle



20.3.3.1. Using the Crevice Output File

The pressure in the top ring land is defined as the cylinder pressure (that is, the pressure in the cells defining the ring landing). Intermediate pressures are available at any point during the ANSYS Fluent session through the `(crevice-summary)` command as previously shown. If the optional data file output is chosen in the `crevice-model-controls`, the intermediate pressures in the defined crevices are printed to the file `crev.out` at the start of each new time step. The format of the file is as follows:

```
  # crank (deg) data-press[0...1...2...3...4...5...6] total_mdot
 1.95500e+02 2.16650e+05 1.01325e+05 1.01325e+05 1.01325e+05 1.01325e+05 1.01325e+05 1.01325e+05 0.0
 1.96000e+02 2.09945e+05 1.06794e+05 1.81553e+05 1.04111e+05 1.48582e+05 1.02202e+05 1.01325e+05 -1.6
 1.96500e+02 2.17787e+05 1.13070e+05 1.88242e+05 1.07960e+05 1.53544e+05 1.03526e+05 1.01325e+05 -1.6
 1.97000e+02 2.17434e+05 1.19065e+05 1.88060e+05 1.11705e+05 1.53475e+05 1.04830e+05 1.01325e+05 -1.6
 1.97500e+02 2.17652e+05 1.24777e+05 1.88299e+05 1.15286e+05 1.53668e+05 1.06081e+05 1.01325e+05 -1.6
 1.98000e+02 2.17937e+05 1.30215e+05 1.88594e+05 1.18711e+05 1.53900e+05 1.07283e+05 1.01325e+05 -1.6
```

where the first column is the current flow time (or crank angle), and the next $n_{cv}+2$ columns are the ring pressures (where n_{cv} is the number of crevice volumes, or $2n_r-1$), including the face pressure on the crevice cell, and the defined pressure at the crevice exit. The final column is the mass flow past the top ring. This file is currently formatted so that it can be read into the free **Gnuplot** plotting package, which is available at `www.gnuplot.info`.

To read the crevice output file into ANSYS Fluent as a data file, you will need to put each column of the crevice output file in its own individual file. The first three lines of each column of the data file should be of the following form:

```
"Title"
"X-Label" "Y-Label"
0 0 0 0
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where the title, x-label, and y-label strings are enclosed by double quotes and the third line of the file contains four zeros. The lines following the first three lines of the file are the columns you want to plot. For example, to plot column 1 versus column 3 of the crevice model output file in ANSYS Fluent, you would enter the following commands in a Linux terminal:

```
cat > crev_col_1_3.dat
"Column 1 vs Column 3"
"Crank Angle (deg)" "Pressure behind ring 1 (Pa)"
0 0 0 0
ctrl-d
```

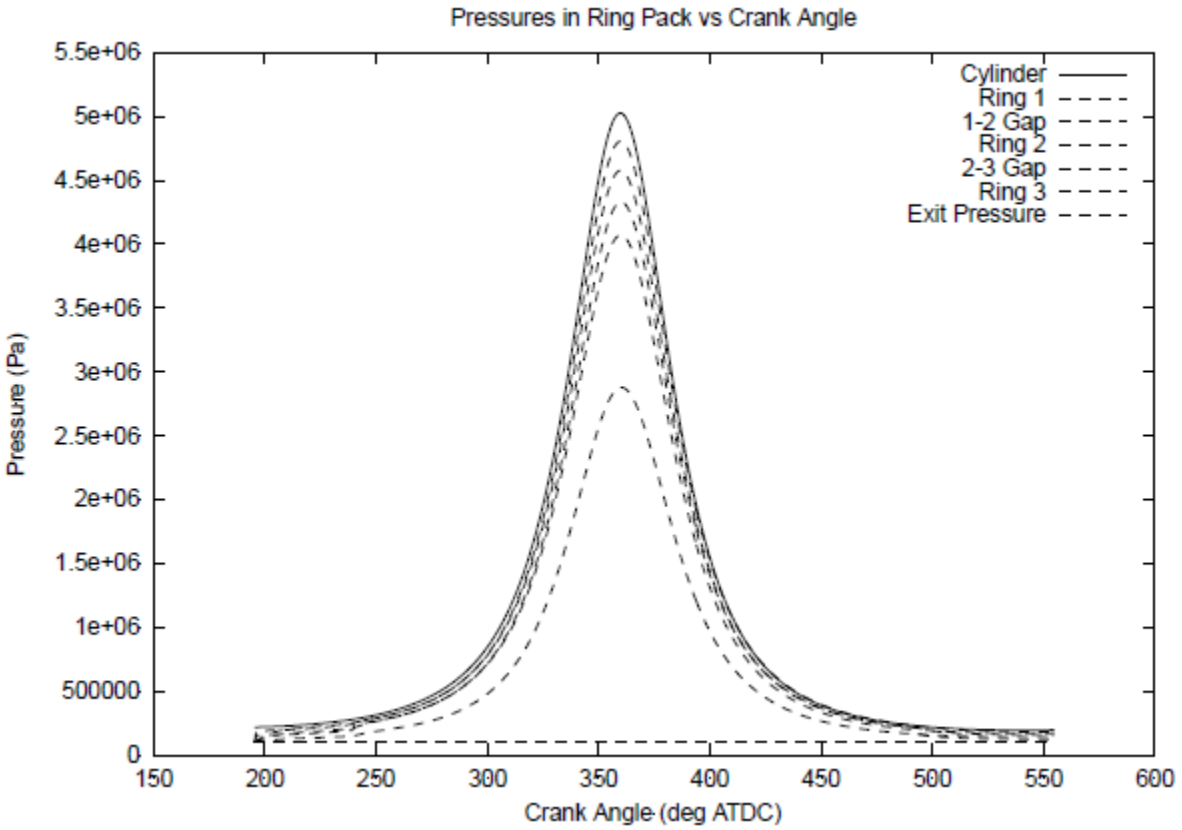
where `ctrl-d` is the end-of-file character made holding down the **Ctrl** key and pressing **d** (**Ctrl**+**d**). To append columns 1 and 3 to this file, enter the following:

```
tail +2 crev.out | awk '{print $1, $3}' >> crev_col_1_3.dat
```

The file `crev_col_1_3.dat` can now be read into ANSYS Fluent using the File XY Plot Dialog Box (p. 2599). See XY Plots of File Data (p. 1844) for details about creating x- y plots. For Windows users, the file `crev.out` can be imported into Excel for plotting purposes without any modification.

A **Gnuplot** plot of the pressure in the ring pack crevices for the above engine simulation is shown in Figure 20.11: Crevice Pressures (p. 1151). After an initial transient period where the flows in the network settle down, Figure 20.11: Crevice Pressures (p. 1151) shows that the pressure in the ring crevices follows the cylinder pressure in form, though with pressure magnitudes that are controlled by the ring pack geometry.

Figure 20.11: Crevice Pressures



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 21: Modeling Pollutant Formation

This chapter discusses how to use the models available in ANSYS Fluent for modeling pollutant formation. For information about the theory behind the models in ANSYS Fluent, see Pollutant Formation in the Theory Guide.

Information is presented in the following sections:

21.1. NOx Formation
21.2. SOx Formation
21.3. Soot Formation
21.4. Using the Decoupled Detailed Chemistry Model

21.1. NOx Formation

The following sections describe how to use the NO_x models in ANSYS Fluent. For information about the theory behind the NO_x models in ANSYS Fluent, see NOx Formation in the Theory Guide.

21.1.1. Using the NOx Model
21.1.2. Solution Strategies
21.1.3. Postprocessing

21.1.1. Using the NOx Model

21.1.1.1. Decoupled Analysis: Overview

NO_x concentrations generated in combustion systems are generally low. As a result, NO_x chemistry has negligible influence on the predicted flow field, temperature, and major combustion product concentrations. It follows that the most efficient way to use the NO_x model is as a postprocessor to the main combustion calculation.

The recommended procedure is as follows:

1. Calculate your combustion problem using ANSYS Fluent as usual.

Important

The premixed combustion model is not compatible with the NO_x model.

Important

If you plan to use the ANSYS Fluent SNCR model for NO_x reduction, you must first include ammonia or urea (depending upon which reagent is employed) as a fluid species in the main combustion calculation and define appropriate ammonia injections, as described later in this section. See Defining the Species in the Mixture (p. 973) for details about adding species to your model and Setting Initial Conditions for the Discrete Phase (p. 1265) for details about creating injections.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Enable the desired NO_x models (thermal, prompt, fuel, and/or N_2O intermediate NO_x, with or without reburn), define the fuel streams (for prompt NO_x and fuel NO_x only), and set the appropriate parameters, as described in this section.

   Setup → Models → Species → NOx Edit...

3. Define the boundary conditions for NO (and HCN, NH_3, or N_2O, if necessary) at flow inlets.

   Setup → Boundary Conditions

4. In the Equations Dialog Box (p. 2511), turn off the solution of all variables except species NO (and HCN, NH_3, or N_2O, based on the model selected).

   Solution → Controls → Equations...

5. Perform calculations until convergence (that is, until the NO—and HCN, NH_3, or N_2O, if solved—species residuals are below 10^{-6}) to ensure that the NO and HCN or NH_3 concentration fields are no longer evolving.

   Solution → Run Calculation

6. Review the mass fractions of NO (and HCN, NH_3, or N_2O) by generating graphical plots or alphanumeric reports in the usual way.

7. Save a new set of case and data files, if desired.

   File → Write → Case & Data...

Inputs specific to the calculation of NO_x formation are explained in the remainder of this section.

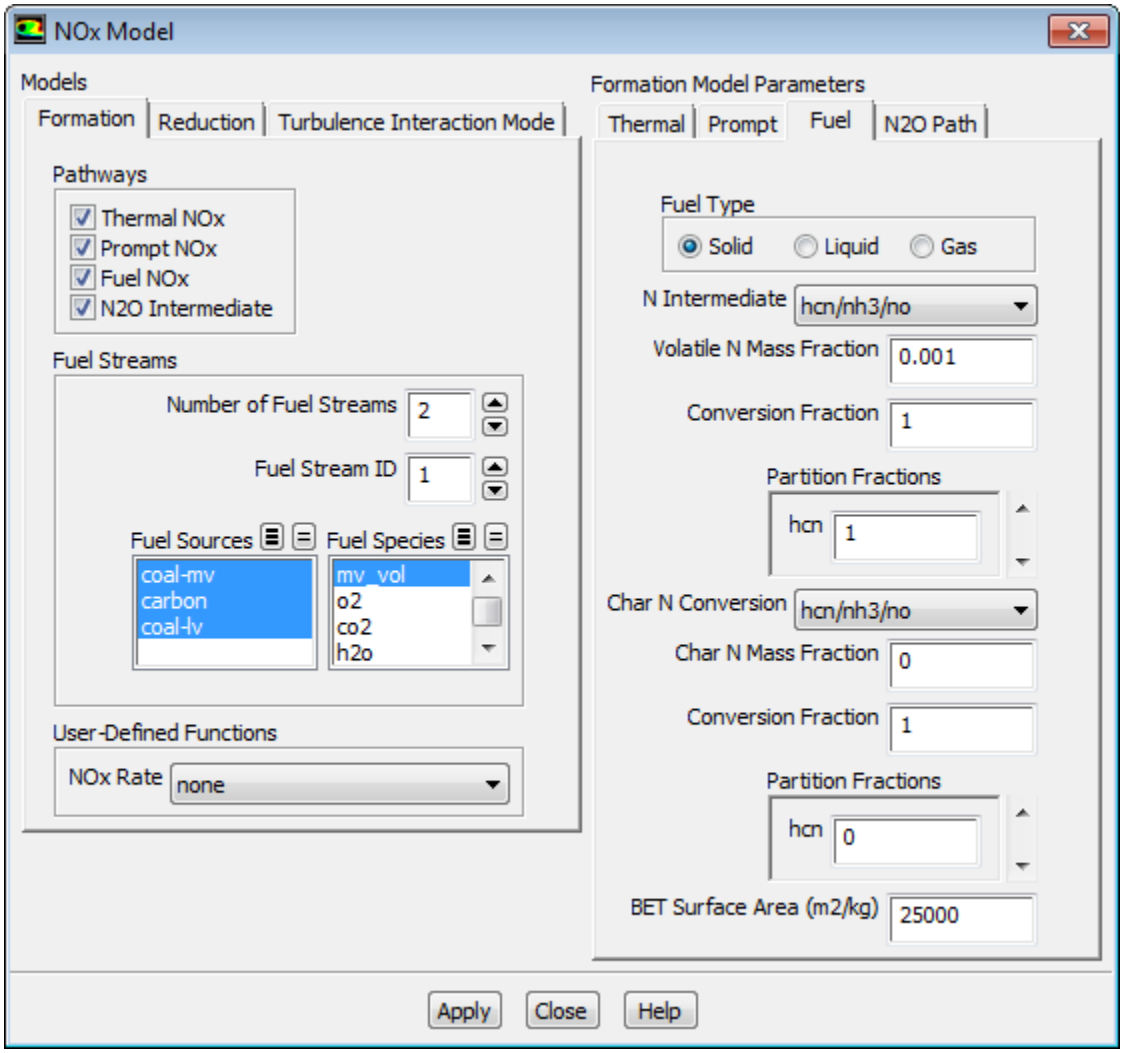
21.1.1.2. Enabling the NOx Models

To enable the NO_x models and set related parameters, you will use the NOx Model Dialog Box (p. 2250) (for example, Figure 21.1: The NOx Model Dialog Box (p. 1155)).

Setup → Models → Species → NOx Edit...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 21.1: The NOx Model Dialog Box



In the **Formation** tab, select the NO_x models under **Pathways** to be used in the calculation of the NO and HCN, NH_3, or N_2O concentrations:

- To enable thermal NO_x, turn on the **Thermal NOx** option.
- To enable prompt NO_x, turn on the **Prompt NOx** option.
- To enable fuel NO_x, turn on the **Fuel NOx** option.

Important

When using the non-premixed combustion model, the following limitation exist for the **Fuel NOx** option:

– The **Fuel NOx** option is only available if the DPM model is also enabled.

- To enable the formation of NO_x through an N_2O intermediate, turn on the **N2O Intermediate** option. (Note that the **N2O Intermediate** option will not appear until you have enabled one of the other NO models listed above.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Your selection(s) under **Pathways** will enable the calculation of thermal, prompt, fuel, and/or N_2O-intermediate NO_x in accordance with the chemical kinetic models described in Thermal NOx Formation through NOx Formation from Intermediate N2O in the Theory Guide. Mean NO formation rates will be computed directly from mean concentrations and temperature in the flow field.

21.1.1.3. Defining the Fuel Streams

When modeling fuel NO_x formation, ANSYS Fluent allows you to define up to three separate fuel streams, and to select the fuel N sources for each fuel stream.

You can define multiple fuel streams to include in your model with the following configurations:

- Solid and liquid fuels (combusting particle and droplet) both contributing to fuel NO_x.
- Two or three solid fuels with different N-content and NO_x model parameters (two or three combusting particles), for example coal blends, coal-biomass cofiring, and so on.
- Two or three liquid fuels with different N-content and NO_x model parameters (two or three droplet or multicomponent particles).
- Gas and solid (or droplet) fuel both contributing to fuel NO_x.

  In addition, you can model one solid fuel contributing to NO_x in the presence of another reacting solid particle not containing any N, for example sorbent injection in a coal furnace, calcination reaction in cement kiln and so on, by specifying the active N fuel source for NO_x formation. For this configuration, you will not need to define multiple fuel streams in your model.

If **Fuel NOx** is enabled in the **Pathways** group box in the **Formation** tab, perform the following steps to define multiple fuel streams:

1. Specify the **Number of Fuel Streams** in the **Fuel Streams** group box.

   Note

   You are allowed up to three separate fuel streams.

2. Define the first fuel stream.

   a. Select the fuel stream to be defined by using the arrow keys of the **Fuel Stream ID** field.

   b. Select the **Fuel Type** in the **Fuel** tab for the **Formation Model Parameters**.

   c. If your **Fuel Type** is **Liquid** or **Solid**, select the N sources from the **Fuel Sources** list.

   - If the **Fuel Type** is **Solid** and you have defined multiple injections with different combusting particle materials in your reacting flow calculation, the available combusting particle materials will be listed in the **Fuel Sources** list. Select one or more materials from the list to be included as fuel sources in the **Fuel NOx** calculation. Your selection will be used to determine the char burnout rate S_c and volatile release rate S_{vol} in the coal fuel NO_x formation rate models described in Fuel NOx from Coal in the Theory Guide. Make sure to deselect all combusting particle materials that do not contribute to NO_x.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If the **Fuel Type** is **Liquid** and you have defined multiple injections with different droplet or multicomponent particle materials in your reacting flow calculation, the available materials will be listed in the **Fuel Sources** list. Select one or more materials from the list to be included as fuel sources in the **Fuel NOx** calculation. Your selection will be used to determine the fuel release rate S_{fuel} in the liquid fuel NO_x formation rate models described in Fuel NOx from Intermediate Hydrogen Cyanide (HCN) and Fuel NOx from Intermediate Ammonia (NH3) in the Theory Guide. Make sure to deselect all droplet and multicomponent materials that do not contribute to NO_x.

d. If you are also modeling **Prompt NOx**, select the fuel species from the **Fuel Species** list. You cannot select more than 5 fuel species for each fuel stream, and the total number of fuel species selected for all the fuel streams combined cannot exceed 10. You can define the same, or different **Fuel Species**, **Fuel Carbon Number** and **Equivalence Ratio** for each stream. The **Fuel Carbon Number** and **Equivalence Ratio** can be modified in the **Prompt** tab under **Formation Model Parameters**.

e. If the **Fuel Type** in your model is **Gas**, select one or more materials from the **Fuel Species** list. Your selection will be used to determine the mean limiting reaction rate $\mathcal{R}_{cf}$ in the gaseous fuel NO_x formation rate models described in Fuel NOx from Intermediate Hydrogen Cyanide (HCN) and Fuel NOx from Intermediate Ammonia (NH3) in the Theory Guide. If you are also modeling **Prompt NOx** formation, the same fuel selection will apply also for the prompt NO_x model.

f. Set the other parameters associated with your selected pathway(s) in the **Prompt** and/or **Fuel** tabs under **Formation Model Parameters**. See Setting Prompt NOx Parameters (p. 1159) and Setting Fuel NOx Parameters (p. 1160) for details.

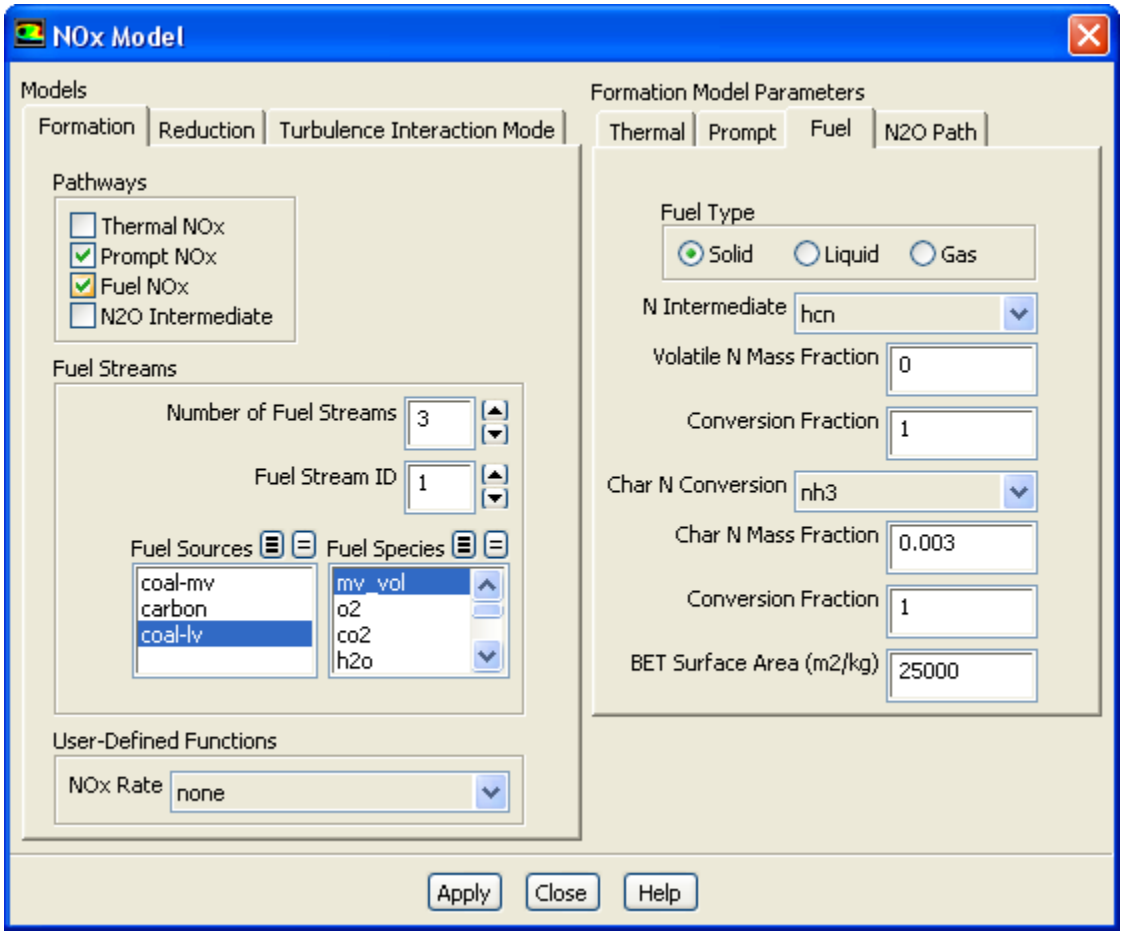
3. Repeat steps 2.(a)–2.(f) for each additional fuel stream.

Important

The gaseous fuel option is available only when the species model is enabled.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 21.2: The NOx Model Dialog Box Displaying the Fuel Streams



Important

Important considerations should be made when reading case and data files set up in a version of ANSYS Fluent 14.5 or earlier:

- When reading a case and data file with multiple injection materials that was set up in a version of ANSYS Fluent previous to and including 14.5, ANSYS Fluent will initialize the injection material specific fuel N sources for the fuel NO_x model. ANSYS Fluent will perform a DPM iteration when the flow iterations are initiated.
- When reading a case file that was set up in a version older than ANSYS Fluent 14.5 with multiple fuel streams defined for the **Fuel NOx** model, you must review the setup and select the N sources from the **Fuel Sources** list.

21.1.1.4. Specifying a User-Defined Function for the NOx Rate

You can choose to specify a user-defined function for the rate of NO_x production. By default, the rate returned from the UDF is added to the rate returned from the standard NO_x production options, if any are selected. You also have the option of replacing any or all of ANSYS Fluent's NO_x rate calculations with your own user-defined NO_x rate.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In addition to or instead of using the UDF to specify the NO_x rate, you can use it to specify custom values for the maximum limit (T_{max}) that is used for the integration of the temperature PDF (when temperature is accounted for in the turbulence interaction modeling).

To use a UDF to add a rate to ANSYS Fluent's NO_x rate calculations, you must compile and load the desired function, and then select it from the **NOx Rate** drop-down list in the **User-Defined Functions** group box in the **Formation** tab. After you have selected the UDF, you have the following options:

- You can specify that your custom rate is added to the ANSYS Fluent NO_x rate calculations, by retaining the default selection of **Add to Fluent Rate** in the **UDF Rate** group box for the appropriate NO_x formation pathway(s) (for example, in the **Fuel** tab).
- You can replace the ANSYS Fluent NO_x rate calculations with your custom rate, by selecting **Replace Fluent Rate** in the **UDF Rate** group box for the appropriate NO_x formation pathway(s) (for example, in the **Fuel** tab).
- You can specify custom values for T_{max}, by selecting **user-defined** from the **Tmax Option** drop-down list in the **Turbulence Interaction Mode** tab.

See the Fluent Customization Manual for details about user-defined functions.

21.1.1.5. Setting Thermal NOx Parameters

The NO_x routines employ three methods for calculation of thermal NO_x (as described in Method 1: Equilibrium Approach in the Theory Guide). You will specify the method to be used in the **Thermal** tab, under **Formation Model Parameters** in the NOx Model Dialog Box (p. 2250):

- To choose the equilibrium method, select **equilibrium** in the **[O] Model** drop-down list.
- To choose the partial equilibrium method, select **partial-equilibrium** in the **[O] Model** or **[OH] Model** drop-down list.
- To use the predicted O and/or OH concentration, select **instantaneous** in the **[O] Model** or **[OH] Model** drop-down list.

Important

Note that the urea model uses the $[OH]$ model.

If you hooked a user-defined function in the **Formation** tab, you can make a selection in the **UDF Rate** group box to specify the treatment of the user-defined NO_x rate:

- Select **Replace Fluent Rate** to replace ANSYS Fluent's thermal NO_x rate calculations with the custom NO_x rate produced by your UDF.
- Select **Add to Fluent Rate** to add the custom NO_x rate produced by your UDF to ANSYS Fluent's thermal NO_x rate calculations.

21.1.1.6. Setting Prompt NOx Parameters

Prompt NO_x formation is predicted using Equation 14.26 and Equation 14.28 in the Theory Guide. For each fuel stream specified in the **Fuel Stream ID** field in the **Formation** tab, set the parameters in the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Prompt tab under **Formation Model Parameters** in the NOx Model Dialog Box (p. 2250) in the following manner:

- Set the **Fuel Carbon Number** to specify the number of carbon atoms per fuel molecule.
- Set the **Equivalence Ratio** as follows:

$$\text{Equivalence Ratio}=\frac{\text{actual fuel-to-air ratio}}{\text{stoichiometric fuel-to-air ratio}} \tag{21.1}$$

If you hooked a user-defined function in the **Formation** tab, you can make a selection in the **UDF Rate** group box to specify the treatment of the user-defined NO_x rate:

- Select **Replace Fluent Rate** to replace ANSYS Fluent's prompt NO_x rate calculations with the custom NO_x rate produced by your UDF.
- Select **Add to Fluent Rate** to add the custom NO_x rate produced by your UDF to ANSYS Fluent's prompt NO_x rate calculations.

21.1.1.7. Setting Fuel NOx Parameters

When using the fuel NO_x model, you must set the parameters in the **Fuel** tab under **Formation Model Parameters** for each fuel stream specified in the **Fuel Stream ID** field in the **Formation** tab.

If you hooked a user-defined function in the **Formation** tab, you can make a selection in the **UDF Rate** group box to specify the treatment of the user-defined NO_x rate:

- Select **Replace Fluent Rate** to replace ANSYS Fluent's fuel NO_x rate calculations with the custom NO_x rate produced by your UDF.
- Select **Add to Fluent Rate** to add the custom NO_x rate produced by your UDF to ANSYS Fluent's fuel NO_x rate calculations.

If there is no NO_x rate UDF or if you selected **Add to Fluent Rate**, you must define fuel parameters. To begin, specify the fuel type in the following manner:

- For solid fuel NO_x, select **Solid** under **Fuel Type**.
- For liquid fuel NO_x, select **Liquid** under **Fuel Type**.
- For gaseous fuel NO_x, select **Gas** under **Fuel Type**.

Note that you can use only one of the fuel types for a given fuel stream. The **Gas** option is available only when the **Species Transport** model is enabled (see Enabling Species Transport and Reactions and Choosing the Mixture Material (p. 960)).

21.1.1.7.1. Setting Gaseous and Liquid Fuel NOx Parameters

If you have selected **Gas** or **Liquid** as the **Fuel Type**, you must also specify the following:

- Select the intermediate species (**hcn**, **nh3**, or **hcn/nh3/no**) in the **N Intermediate** drop-down list.
- Set the correct mass fraction of nitrogen in the fuel (kg nitrogen per kg fuel) in the **Fuel N Mass Fraction** field.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Specify the overall fraction of the fuel N, by mass, that will be converted to the intermediate species and/or product NO in the **Conversion Fraction** field. The **Conversion Fraction** for the **N Intermediate** has a default value of 1. Therefore, any remaining N will not contribute to NO_x formation. This is based on the assumption that the remaining volatile N will convert to gas phase nitrogen. However, this has very little effect on the overall mass fraction of gas phase nitrogen. Therefore, you do not have to solve for nitrogen species when solving pollutant transport equations.

- If you selected **hcn/nh3/no** as the intermediate, specify the fraction of the converted fuel N, by mass, that will become **hcn** and **nh3** under **Partition Fractions**. The fraction of fuel N that will become NO will be calculated by the remainder.

  Note that setting a partition fraction of 0 for both HCN and NH_3 is equivalent to assuming that all fuel N is converted to the final product NO, whereas a partition fraction of 0 for HCN and 1 for NH_3 is the same as selecting **nh3** as the intermediate.

ANSYS Fluent will use Equation 14.31 and Equation 14.32 in the Theory Guide (for HCN) or Equation 14.42 and Equation 14.43 in the Theory Guide (for NH_3) to predict NO formation for a gaseous or liquid fuel.

Important

Note that there is a limitation that must be considered when defining more than one liquid fuel stream. See Defining the Fuel Streams (p. 1156) for details.

21.1.1.7.2. Setting Solid (Coal) Fuel NOx Parameters

For solid (coal) fuel, ANSYS Fluent will use Equation 14.55 and Equation 14.56 in the Theory Guide (for HCN) or Equation 14.62 and Equation 14.63 in the Theory Guide (for NH_3) to predict NO formation. Several inputs are required for the coal fuel NO_x model as follows:

- Select the intermediate species (**hcn**, **nh3**, or **hcn/nh3/no**) in the **N Intermediate** drop-down list.

- Specify the mass fraction of nitrogen in the volatiles in the **Volatile N Mass Fraction** field.

- Specify the overall fraction of the volatile N, by mass, that will be converted to the intermediate species and/or product NO in the **Conversion Fraction** field.

- If you selected **hcn/nh3/no** as the volatile N intermediate, specify the fraction of the converted volatile N, by mass, that will become **hcn** and **nh3** under **Partition Fractions**. The fraction of volatile N that will become NO will be calculated by the remainder.

- Select the char N conversion path from the **Char N Conversion** drop-down list as **no**, **hcn**, **nh3**, or **hcn/nh3/no**. Note that **hcn** or **nh3** can be selected only if the same species has been selected as the intermediate species in the **N Intermediate** drop-down list.

- Specify the mass fraction of nitrogen in the char in the **Char N Mass Fraction** field.

- Specify the overall fraction of the char N, by mass, that will be converted to the intermediate species and/or product NO in the **Conversion Fraction** field.

- If you selected **hcn/nh3/no** as the char N conversion, specify the fraction of the converted char N, by mass, that will become **hcn** and **nh3** under **Partition Fractions**. The fraction of char N that will become NO will be calculated by the remainder.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Define the BET internal pore surface area (see BET Surface Area in the Theory Guide for details) of the particles in the **BET Surface Area** field.

Important

Note that there are limitations that must be considered when defining more than one solid fuel stream. See Defining the Fuel Streams (p. 1156) for details.

The following equations are used to determine the mass fraction of nitrogen in the volatiles and char:

$$\dot{m}_{N_{v/c}} = \dot{m}_{v/c} * mf_{N_{v/c}} \tag{21.2}$$

where

$\dot{m}_{N_{v/c}}$ = rate of release of fuel nitrogen in kg/s

$\dot{m}_{v/c}$ = rate of release of volatiles (v) or char (c) in kg/s

$mf_{N_{v/c}}$ = mass fraction of nitrogen in volatiles or char

Let

TN_{fuel} = total nitrogen mass fraction in daf coal (that is, from daf ultimate analysis)

N_{split} = char nitrogen as a fraction of total nitrogen

F_{vol} = mass fraction of volatiles in daf coal

F_{char} = mass fraction of char in daf coal

Then the following should hold:

$$F_{vol} + F_{char} = 1 \tag{21.3}$$

$$\frac{F_{char} * mf_{N_c}}{TN_{fuel}} = N_{split} \tag{21.4}$$

$$F_{vol} * mf_{N_v} + F_{char} * mf_{N_c} = TN_{fuel} \tag{21.5}$$

$$mf_{N_v} = \left(1 - N_{split}\right) * \frac{TN_{fuel}}{F_{vol}} \tag{21.6}$$

$$mf_{N_c} = N_{split} * \frac{TN_{fuel}}{F_{char}} \tag{21.7}$$

21.1.1.8. Setting N2O Pathway Parameters

The formation of NO through an N_2O intermediate can be predicted by two methods. You will specify the method to be used in the **N2O Path** tab.

- To choose the quasi-steady-state method, select **quasi-steady** in the **N2O Model** drop-down list.

Important

The transport equation for the species N_2O will not be solved for N_2O. However, N_2O will be updated at every iteration. Therefore, the residual values that appear for N_2O are always zero. Do not be alarmed if the solver keeps printing zero at each iteration.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To choose the simplified form of the N_2O-intermediate mechanism, select **transported-simple** in the **N2O Model** drop-down list. Here, the species N_2O is added to the list of pollutant species, and its mass fraction is solved via a transport equation.

The atomic O concentration will be calculated according to the thermal NO_x **[O] Model** that you have specified previously. If you have not selected the **Thermal NOx** pathway, then you will be given the option to specify an **[O] Model** for the N_2O pathway calculation. The same three options for the thermal NO_x **[O] Model** will be the available options.

If you hooked a user-defined function in the **Formation** tab, you can make a selection in the **UDF Rate** group box to specify the treatment of the user-defined NO_x rate:

- Select **Replace Fluent Rate** to replace the NO_x rate calculated by ANSYS Fluent using N_2O intermediates with the custom NO_x rate produced by your UDF.
- Select **Add to Fluent Rate** to add the custom NO_x rate produced by your UDF to the NO_x rate calculated by ANSYS Fluent using N_2O intermediates.

21.1.1.9. Setting Parameters for NOx Reburn

To enable NO_x reduction by reburning, click the **Reduction** tab in the NOx Model Dialog Box (p. 2250) and enable the **Reburn** option under **Methods**. In the expanded portion of the dialog box, as shown in Figure 21.3: The NOx Dialog Box Displaying the Reburn Reduction Method (p. 1164), click the **Reburn** tab under **Reduction Method Parameters**, where you can choose from the following options:

- To choose the instantaneous method, select **instantaneous [CH]** in the **Reburn Model** drop-down list.

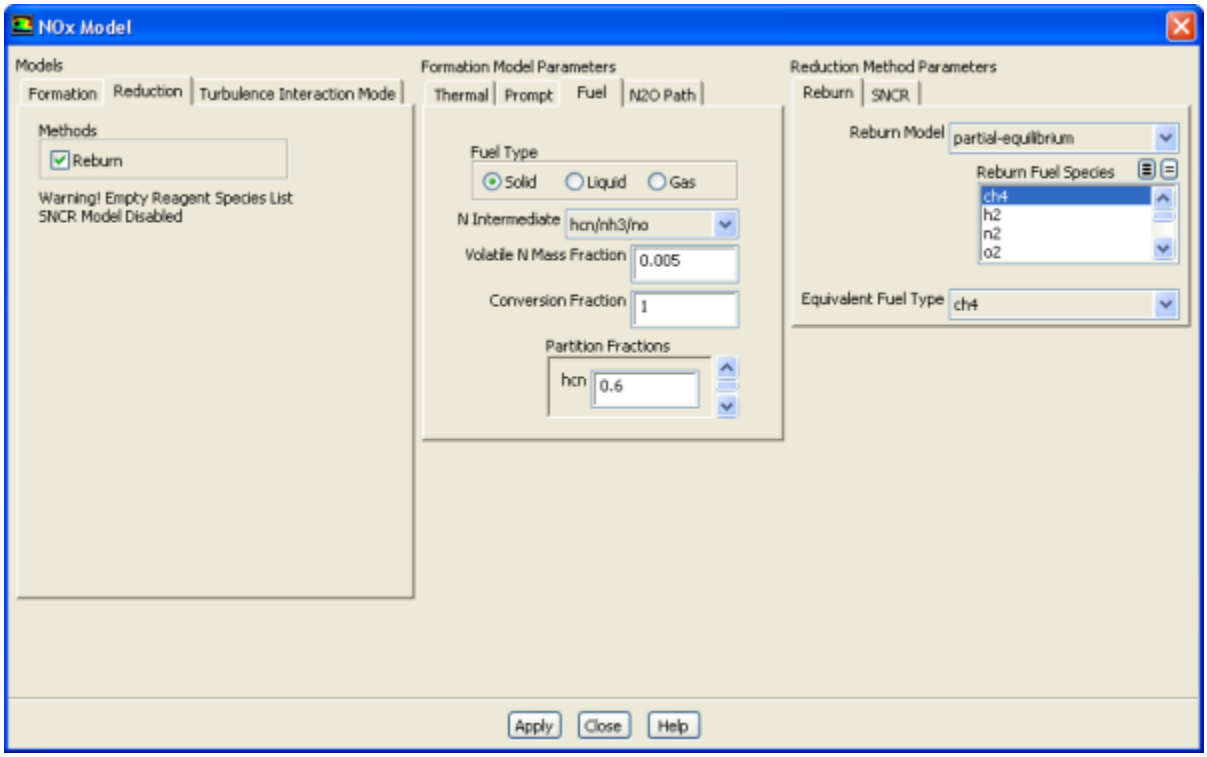
  Important

  When you use this method, you must be sure to include the species CH, CH_2, and CH_3 in your problem definition. See NOx Reduction by Reburning in the Theory Guide for details.

- To choose the partial equilibrium method, select **partial-equilibrium** in the **Reburn Model** drop-down list. You then must select the **Reburn Fuel Species** from the list of available species. ANSYS Fluent will allow you select up to five reburn fuel species. Specify the **Equivalent Fuel Type** (**ch4**, **ch3**, **ch2**, or **ch**). For example, if you choose methane as the reburn fuel, then the **Equivalent Fuel Type** would be **ch4**. If you choose a reburn fuel such as **hv_vol** (a volatile component of coal), then you must specify the most appropriate equivalent hydrocarbon fuel type so that the partial equilibrium model will be set up correctly.
- Due to coal volatiles behaving very differently, it is important to select the correct equivalent fuel type. You must first consider the volatile fuel composition, then check the C/H ratio to find the fuel that most closely matches CH, CH_2, CH_3, or CH_4 [60] (p. 2890). While the method for best determining the equivalent fuel is debatable, considering the C/H ratio of the fuel itself is a reasonable indicator.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 21.3: The NOx Dialog Box Displaying the Reburn Reduction Method



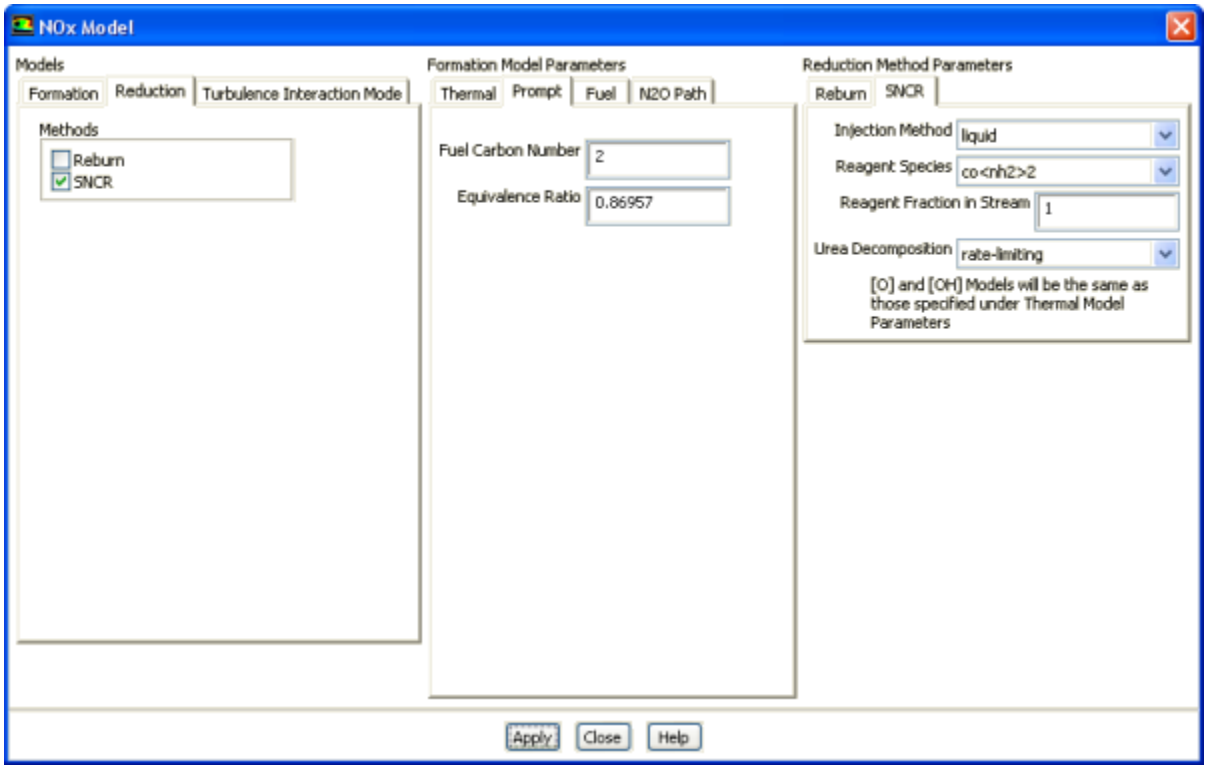
21.1.1.10. Setting SNCR Parameters

Prior to enabling reduction by SNCR, make sure that you have included in the species list **nh3** (for reduction by ammonia injection) and **co<nh2>2** (for reduction by urea injection). See NOx Reduction by SNCR in the Theory Guide for detailed information about SNCR theory.

To enable NO_x reduction by SNCR, click the **Reduction** tab in the NOx Model Dialog Box (p. 2250) and enable the **SNCR** option under **Methods**, as shown in Figure 21.4: The NOx Dialog Box Displaying the SNCR Reduction Method (p. 1165).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 21.4: The NOx Dialog Box Displaying the SNCR Reduction Method



Then click the **SNCR** tab under **Reduction Method Parameters**, where you can choose from the following options:

- To have ammonia or urea included as a gas-phase pollutant species from the injection locations, select **gaseous** in the **Injection Method** drop-down list.

  If you plan to select this option for NO_x postprocessing, then you must also include ammonia or urea as a gas-phase species. Additionally, you must specify the mass fraction of ammonia or urea at the respective inlet for the SNCR injection. You must include this set of inputs prior to the main ANSYS Fluent combustion calculation.

- To have ammonia included as a liquid-phase pollutant species from the injection locations, select **liquid** in the **Injection Method** drop-down list. If urea is injected as a liquid solution, then select **liquid** in the **Injection Method** drop-down list. Note that you must enable the DPM model with urea or ammonia included as a material.

  Note

  All particle types such as droplets, combusting, and multicomponent particles are compatible with the SNCR model.

  If you plan to select this option for NO_x postprocessing, then you must include NH_3 as both a gas-phase and liquid-phase species. Additionally, you must specify injection locations for liquid droplet ammonia particles and set gaseous ammonia as the evaporation species. You must include this set of inputs in conjunction with the main ANSYS Fluent combustion calculation.

  Since urea is a subliming solid, and usually is injected as a solution, mixed in water, you have to define solid properties for urea under the Create/Edit Materials Dialog Box (p. 2302). It is assumed that

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the water evaporates before urea begins its subliming process. The sublimation process is modeled similar to the single rate devolatilization model of coal. You will supply the value for the sublimation rate (s^{-1}). You must specify the water content while defining the injection properties.

- Specify the SNCR **Reagent Species** as **nh3** (ammonia) or **co<nh2>2** (urea) in the drop-down list.
- When using the non-premixed combustion model with a liquid-phase reagent injection, enter a value in the **Reagent Fraction in Stream** to specify the mass fraction of the reagent in the reagent stream. The remaining mass fraction is assumed to be water. If you enabled a secondary stream in your PDF calculation, by default the secondary stream will act as the reagent stream. You can assign the primary stream as the reagent stream by using the text command that follows (enter `0` in response to the `PDF Stream ID` prompt that follows the `Injection Method` prompt):

  ```
  define/models/nox-parameters/nox-chemistry
  ```

- If the **Reagent Species** selected is **co <nh2> 2**, then you will either accept the **rate-limiting** option for **Urea Decomposition**, or specify the **NH3 Conversion** value when selecting a **user-specified Urea Decomposition**.

  You will use the urea decomposition under the **SNCR** tab to define which of the two decomposition models is to be used. The first model (which is the default) is the rate-limiting decomposition model, as given in Table 14.3: Two-Step Urea Breakdown Process in the Theory Guide. ANSYS Fluent will then calculate the source terms according to the rates given in Table 14.3: Two-Step Urea Breakdown Process in the Theory Guide. The second model is for when you assume that the urea decomposes instantly into ammonia and $HNCO$ at a given proportion. In this case, you will specify the molar conversion fraction for ammonia, assuming that the rest of the urea is converted to $HNCO$. An example value is given above.

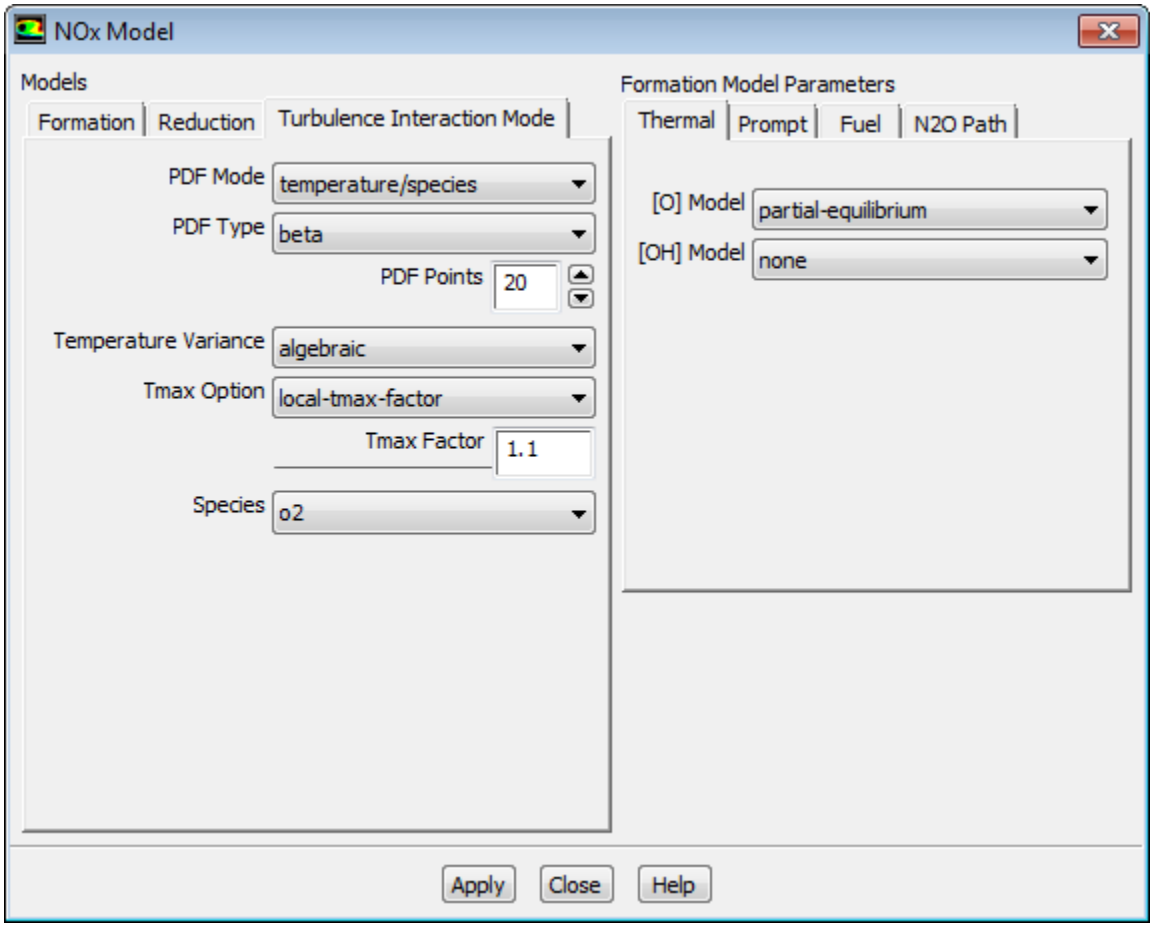
  The value for **user-specified** NH_3 conversion is the mole fraction of NH_3 in the mixture of NH_3 and $HNCO$ that is instantly created from the reagent injection. In this case, there is no urea source because all of reagent is assumed to convert to both NH_3 and $HNCO$, instantly.

21.1.1.11. Setting Turbulence Parameters

If you want to take into account turbulent fluctuations (as described in NOx Formation in Turbulent Flows in the Theory Guide) when you compute the specified NO_x formation (thermal, prompt, and/or fuel, with or without reburn), define the turbulence parameters in the **Turbulence Interaction Mode** tab.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 21.5: The NOx Model Dialog Box and the Turbulence Interaction Mode Tab



Select one of the options in the **PDF Mode** drop-down list in the **Turbulence Interaction Mode** tab:

- Select **temperature** to take into account fluctuations of temperature.
- Select **temperature/species** to take into account fluctuations of temperature and mass fraction of the species selected in the **Species** drop-down list (which appears when you select this option).
- (non-premixed and partially premixed combustion calculations only) Select **mixture fraction** to take into account fluctuation in the mixture fraction(s).

Important

When modeling the formation of other pollutants along with NO_x, you should compare the selections made in the **PDF Mode** drop-down lists in the **Turbulence Interaction Mode** group boxes of the NOx Model Dialog Box (p. 2250) and the **Turbulence Interaction Mode** group boxes of the **SOx Model** and **Soot Model** dialog boxes. If **mixture fraction** is selected in any of these dialog boxes, then it must be selected in all of the others as well.

The **mixture fraction** option is available only if you are using either the non-premixed or partially pre-mixed combustion model to model the reacting system. If you use the **mixture fraction** option, the instantaneous temperatures and species concentrations are taken from the PDF look-up table as a function of mixture fraction and enthalpy and the instantaneous NO_x rates are calculated at each cell. The PDF used for convoluting the instantaneous NO_x rates is the same as the one used to compute the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

mean flow-field properties. For example, for single-mixture fraction models the beta PDF is used, and for two-mixture fraction models, the beta or the double delta PDF can be used. The PDF for mixture fraction is calculated from the values of mean mixture fraction and variance at each cell, and the instantaneous NO_x rates are convoluted with the mixture fraction PDF to yield the mean rates in turbulent flow.

Note

When `mixture fraction` is selected as **PDF mode**, ANSYS Fluent calculates and tabulates the NO_x rates apriori for faster computations of NO_x rates. The NO_x rates are tabulated at the first NO_x iteration and a message

```
Performing Pollutant PDF integrations...
```

is displayed in the Fluent console.

If you selected **temperature** or **temperature/species** for the **PDF Mode**, you should define the following parameters in the **Turbulence Interaction Mode** tab:

PDF Type
allows you to specify the shape of the PDF, which is then integrated to obtain mean rates for the temperature and (if you selected **temperature/species** for the **PDF Mode**) the species. If you select **beta**, the PDF will be modeled using Equation 14.108 in the Theory Guide. If you select **gaussian**, the PDF will be modeled using Equation 14.111 in the Theory Guide.

PDF Points
allows you to specify the number of points used to integrate the beta or Gaussian function in Equation 14.105 or Equation 14.106 in the Theory Guide on a histogram basis. The default value of 10 is a minimum value. It is recommended that you run the calculation with this minimum value to convergence, and then keep increasing the value (for example, 20–25) until the pollutants of interest stop changing. Increasing this value may improve accuracy, but will also increase the computation time.

Temperature Variance
allows you to specify the form of transport equation that is solved to calculate the temperature variance. The default selection is **algebraic**, which is an approximate form of the transport equation (see Equation 14.114 of the Theory Guide). You have the option of selecting **transported** to instead solve Equation 14.113 in the Theory Guide. Though the **transported** form is more exact, it is also more expensive computationally.

Tmax Option
provides various options for determining the maximum limit of the temperature used for the integration of the PDF (see Equation 8.43 in the *Fluent Theory Guide*) to calculate mean rates for turbulent fluctuations of NO_x:

- The default selection is **global-tmax**, which sets the limit as the maximum temperature in the flow field.
- You can select **local-tmax** if you would rather obtain cell-based maximum temperature limits by multiplying the local cell mean temperature by the value entered in **Tmax Factor**.
- You can select **specified-tmax** to set the limit for each cell to be the value entered in **Tmax**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you have selected a user-defined function in the **NOx Rate** drop-down list in the **Formation** tab, then you can select **user-defined** so that the temperature limit is specified by a user-defined function. See the Fluent Customization Manual for details about user-defined functions.

Species

only appears if you have selected **temperature/species** for the **PDF Mode**. Your selection in this drop-down menu determines which species' mass fraction is included in the NO_x formation calculations.

Important

Note that the species variance will always be calculated using the algebraic form of the transport equation (Equation 14.114 in the Theory Guide).

21.1.1.12. Defining Boundary Conditions for the NOx Model

At flow inlet boundaries, you must specify the **Pollutant NO Mass Fraction**, and if necessary, the **Pollutant HCN Mass Fraction**, **Pollutant NH3 Mass Fraction**, and **Pollutant N2O Mass Fraction**.

Setup → **Boundary Conditions**

You can retain the default inlet values of zero for these quantities or you can enter nonzero numbers as appropriate for your combustion system.

21.1.2. Solution Strategies

To solve for NO_x products, perform the following steps:

- (optional) If the discrete phase model (DPM) is turned on (by enabling the **Interaction with Continuous Phase**) to run with the NO_x model, then set the **DPM Iteration Interval** to `0` such that no DPM iterations are performed as the NO_x case is being solved.
- In the Equations Dialog Box (p. 2511) of the Solution Controls Task Page (p. 2508), turn off the solution of all variables except species NO (and HCN, NH_3, or N_2O, based on the model selected).

  Solution → **Controls** → **Equations...**
- In the **Solution Controls** task page, set a suitable value for the NO (and HCN, NH_3, or N_2O, if appropriate) under-relaxation. A value of 1.0 is suggested, although lower values may be required for certain problems (that is, if convergence cannot be obtained, try a lower under-relaxation value).
- In the Residual Monitors Dialog Box (p. 2789), decrease the convergence criterion for NO (and HCN, NH_3, or N_2O, if appropriate) to 10^{-6}.

  Solution → **Monitors** → **Residuals** **Edit...**
- Perform calculations until convergence (that is, until the NO—and HCN, NH_3, or N_2O, if solved—species residuals are below 10^{-6}) to ensure that the NO and HCN or NH_3 concentration fields are no longer evolving.

  Solution → **Run Calculation**

21.1.3. Postprocessing

When you compute NO_x formation, the following additional variables will be available for postprocessing:

- **Mass fraction of Pollutant no**
- **Mass fraction of Pollutant hcn** (appropriate fuel NO_x model only)
- **Mass fraction of Pollutant nh3** (appropriate fuel NO_x model only)
- **Mass fraction of Pollutant n2o** (N_2O-intermediate model only)
- **Mass fraction of Pollutant urea** (SNCR urea injection)
- **Mass fraction of Pollutant hnco** (SNCR urea injection)
- **Mass fraction of Pollutant nco** (SNCR urea injection)
- **Mole fraction of Pollutant no**
- **Mole fraction of Pollutant hcn** (appropriate fuel NO_x model only)
- **Mole fraction of Pollutant nh3** (appropriate fuel NO_x model only)
- **Mole fraction of Pollutant n2o** (N_2O-intermediate model only)
- **Mole fraction of Pollutant urea** (SNCR urea injection)
- **Mole fraction of Pollutant hnco** (SNCR urea injection)
- **Mole fraction of Pollutant nco** (SNCR urea injection)
- **no Density**
- **hcn Density** (appropriate fuel NO_x model only)
- **nh3 Density** (appropriate fuel NO_x model only)
- **n2o Density** (N_2O-intermediate model only)
- **urea Density** (SNCR urea injection)
- **hnco Density** (SNCR urea injection)
- **nco Density** (SNCR urea injection)
- **Rate of no** (from the individual pathways)
- **Rate of hcn** (appropriate fuel NO_x model only)
- **Rate of nh3** (appropriate fuel NO_x model only)
- **Rate of n2o** (N_2O-intermediate model only)
- **Rate of urea** (SNCR urea injection)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Rate of hnco** (SNCR urea injection)
- **Rate of nco** (SNCR urea injection)

These variables are contained in the **NOx...** category of the variable selection drop-down list that appears in postprocessing dialog boxes. Additional NO rates from individual pathways, **Thermal**, **Prompt**, **Fuel**, **N2O Path**, and **SNCR** can be plotted.

21.2. SOx Formation

The following sections describe how to use the SO_x model in ANSYS Fluent. For information about the theory behind the SO_x models in ANSYS Fluent, see SOx Formation in the Theory Guide.

21.2.1. Using the SOx Model
21.2.2. Solution Strategies
21.2.3. Postprocessing

21.2.1. Using the SOx Model

When the sulfur content in the fuel is low, SO_x concentrations that are generated in combustion generally have minimal influence on the predicted flow field, temperature, and major combustion product concentrations. The most efficient way to use the SO_x model is as a postprocessor to the main combustion calculation. However, if the sulfur content is high, then SO_x formation should be coupled with the gas phase combustion process rather than treating it as a postprocessing step.

The procedure for enabling and setting up the model for a decoupled solution is as follows:

1. Calculate your combustion problem using ANSYS Fluent.

   Important

   The premixed combustion model and the Flamelet generated manifold model are not compatible with the SO_x model.

2. Enable the SO_x model, define the fuel streams, and set the appropriate parameters, as described in this section.

   Setup → **Models** → **Species** → **SOx** **Edit...**

3. Define the boundary conditions for SO_2 and H_2S (and SO_3, SH, or SO if necessary) at flow inlets.

   Setup → **Boundary Conditions**

4. In the Equations Dialog Box (p. 2511), turn off the solution of all variables except species SO_2 and H_2S (and SO_3, SH, or SO, based on your selections).

   Solution → **Controls** → **Equations...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Perform calculations until convergence (that is, until the SO_2 and H_2S—and SO_3, SH, or SO, if solved—species residuals are below 10^{-6}) to ensure that the SO_2 and H_2S concentration fields are no longer evolving.

   Solution → **Run Calculation**

6. Review the mass fractions of SO_2 and H_2S (and SO_3, SH, or SO) by generating graphical plots or alphanumeric reports in the usual way.

7. Save a new set of case and data files, if desired.

   File → **Write** → **Case & Data...**

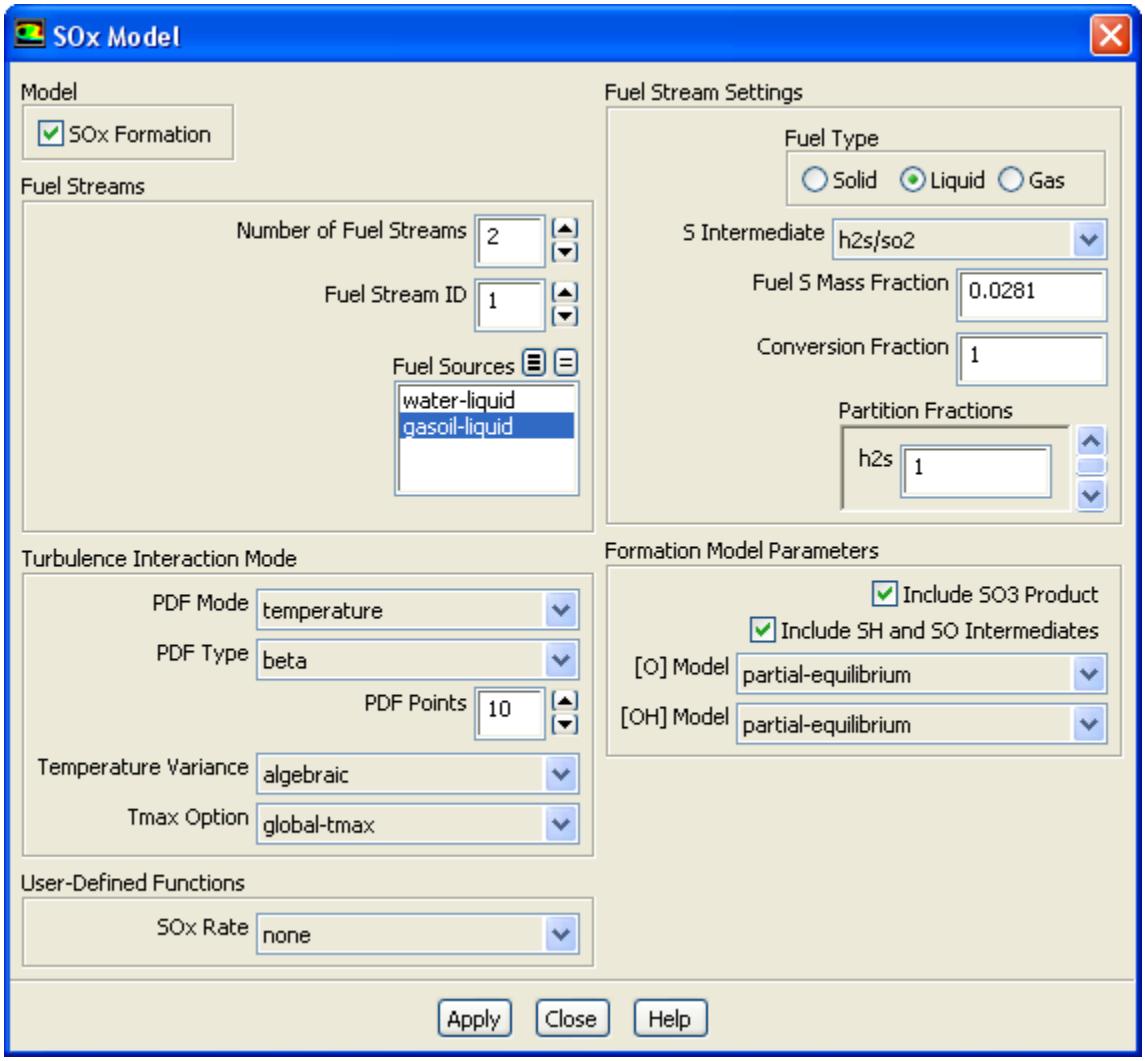
21.2.1.1. Enabling the SOx Model

To model SO_x formation, enable the **SOx Formation** option in the SOx Model Dialog Box (p. 2258) (Figure 21.6: The SOx Model Dialog Box (p. 1173)).

Setup → **Models** → **Species** → **SOx** **Edit...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 21.6: The SOx Model Dialog Box



21.2.1.2. Defining the Fuel Streams

When modeling fuel SO_x formation, ANSYS Fluent allows you to define up to three separate fuel streams, and to select the fuel S sources for each fuel stream.

You can define multiple fuel streams to include in your model with the following configurations:

- Solid and liquid fuels (combusting particle and droplet) both contributing to fuel SO_x.
- Two or three solid fuels with different S-content and SO_2 /H_2S production models (two or three combusting particles), for example coal blends, coal-biomass cofiring, and so on.
- Two or three liquid fuels with different S-content and SO_2 /H_2S production models (two or three droplet or multicomponent particles).
- Gas and solid (or droplet) fuel both contributing to fuel SO_x.

  In addition, you can model one solid fuel contributing to SO_x in the presence of another reacting solid particle not containing any S, by specifying the active S fuel source for SO_x formation. For this configuration, you will not need to define multiple fuel streams in your model.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can define multiple fuel streams when you are modeling SO_x formation, as shown in the following steps:

1. Specify the **Number of Fuel Streams** in the **Fuel Streams** group box. You are allowed up to three separate fuel streams.

2. Define the first fuel stream.

   a. Select the fuel stream to be defined by using the arrow keys of the **Fuel Stream ID** field.

   b. Select the **Fuel Type** in the **Fuel Stream Settings** group box.

   c. If your **Fuel Type** is **Liquid** or **Solid**, select the S sources from the **Fuel Sources** list.

      - If the **Fuel Type** is **Solid** and you have defined multiple injections with different combusting particle materials in your reacting flow calculation, the available combusting particle materials will be listed in the **Fuel Sources** list. Select one or more materials from the list to be included as fuel sources in the **SOx** calculation. Your selection will be used to determine the char burnout rate S_c and volatile release rate S_{vol} in Equation 14.124 and Equation 14.125 in the Theory Guide. Make sure to deselect all combusting particle materials that do not contribute to SO_x.

      - If the **Fuel Type** is **Liquid** and you have defined multiple injections with different droplet or multicomponent particle materials in your reacting flow calculation, the available materials will be listed in the **Fuel Sources** list. Select one or more materials from the list to be included as fuel sources in the **SOx** calculation. Your selection will be used to determine the fuel release rate S_{fuel} in Equation 14.123 in the Theory Guide. Make sure to deselect all droplet and multicomponent materials that do not contribute to SO_x.

   d. If the **Fuel Type** in your model is **Gas**, select one or more materials from the **Fuel Species** list. You cannot select more than 5 fuel species for each fuel stream, and the total number of fuel species selected for all the fuel streams combined cannot exceed 10. Your selection will be used to determine the mean limiting reaction rate $\mathcal{R}_{cf}$ in Equation 14.122 in the Theory Guide.

   e. Specify the parameters for this particular fuel stream in the **Fuel Stream Settings** group box. See Defining the SOx Fuel Stream Settings (p. 1175) for details.

3. Repeat steps 2.(a)–2.(e) for each additional fuel stream.

Important

Important considerations should be made when reading case and data files set up in a version of ANSYS Fluent 14.5 or earlier:

- When reading a case and data file with multiple injection materials that was set up in a version of ANSYS Fluent previous to and including 14.5, ANSYS Fluent will initialize the injection material specific fuel S sources for the fuel SO_x model. ANSYS Fluent will perform a DPM iteration when the flow iterations are initiated.

- When reading a case file that was set up in a version older than ANSYS Fluent 14.5 with multiple fuel streams defined for the **SOx** model, you must review the setup and select the S sources from the **Fuel Sources** list.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

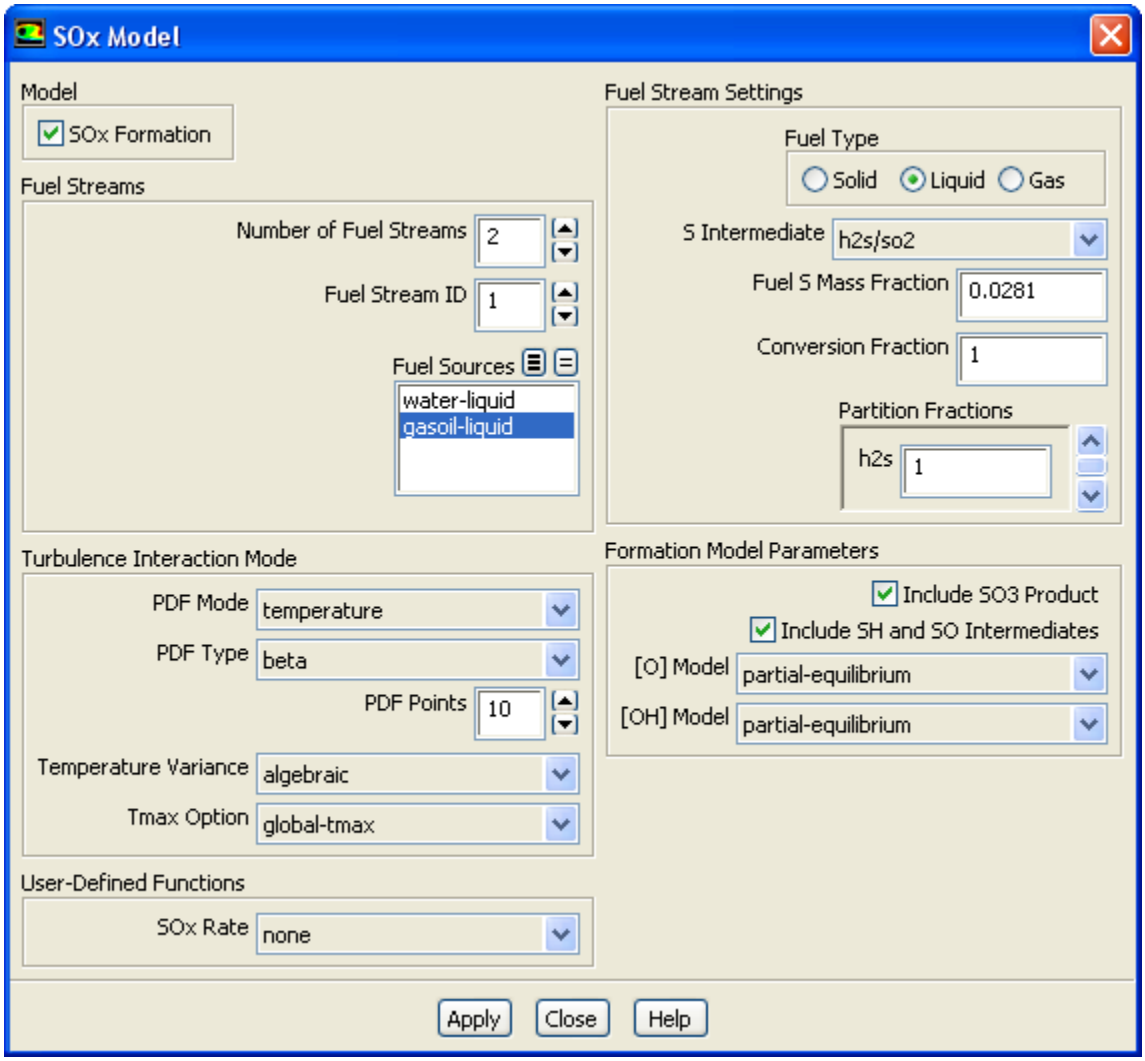
21.2.1.3. Defining the SOx Fuel Stream Settings

When using the SO_x model, you must set the parameters in the **Fuel Stream Settings** group box for each fuel stream specified in the **Fuel Stream ID** field.

To begin, specify the fuel type in the following manner:

- To calculate SO_x formation from a solid fuel, select **Solid** under **Fuel Type**.
- To calculate SO_x formation from a liquid fuel, select **Liquid** under **Fuel Type**.
- To calculate SO_x formation from a gaseous fuel, select **Gas** under **Fuel Type**.

Figure 21.7: The SOx Model Dialog Box Displaying Liquid Fuel Parameters



Note that you can use only one of the fuel types for a given fuel stream. The **Gas** option is available only when the **Species Transport** model is enabled (see Enabling Species Transport and Reactions and Choosing the Mixture Material (p. 960)).

21.2.1.3.1. Setting SOx Parameters for Gaseous and Liquid Fuel Types

If you have selected **Gas** or **Liquid** as the **Fuel Type**, you must specify the following:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select the intermediate species (**h2s**, **so2**, or **h2s/so2**) in the **S Intermediate** drop-down list.
- Set the correct mass fraction of sulfur in the fuel (kg sulfur per kg fuel) in the **Fuel S Mass Fraction** field.
- Specify the overall fraction of the fuel S, by mass, that will be converted to the intermediate species and/or product SO_2 in the **Conversion Fraction** field. Therefore, any remaining S will not contribute to SO_x formation. This is based on the assumption that the remaining volatile S will convert to gas phase sulfur. The **Conversion Fraction** for the **S Intermediate** has a default value of 1.
- If you selected **h2s/so2** as the intermediate, you must set the fraction of the converted fuel S, by mass, that will become **h2s** under **Partition Fractions**. The fraction of fuel S that will become SO_2 will be calculated by the remainder.

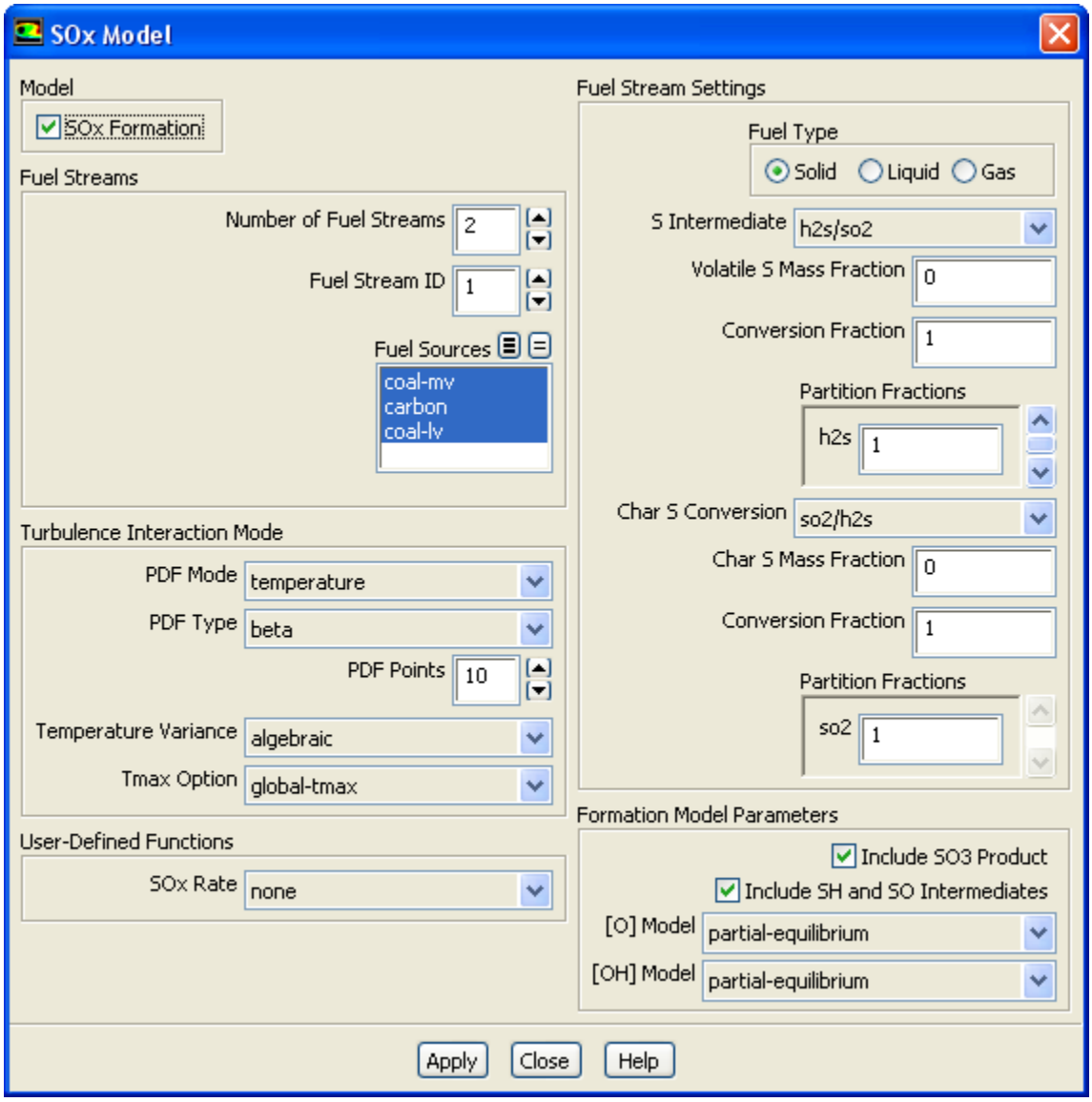
  Note that setting a partition fraction of 0 for H_2S is equivalent to assuming that all fuel S is converted to the final product SO_2.

21.2.1.3.2. Setting SOx Parameters for a Solid Fuel

For solid fuel, several inputs are required for the SO_x model.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 21.8: The SOx Model Dialog Box Displaying Solid Fuel Parameters



- Select the intermediate species (**h2s**, **so2**, or **h2s/so2**) in the **S Intermediate** drop-down list.
- Specify the mass fraction of sulfur in the volatiles in the **Volatile S Mass Fraction** field.
- Specify the overall fraction of the volatile S, by mass, that will be converted to the intermediate species and/or product SO_2 in the **Conversion Fraction** field.
- If you selected **h2s/so2** as the volatile S intermediate, you must specify the fraction of the converted volatile S, by mass, that will become **h2s** under **Partition Fractions**. The fraction of volatile S that will become SO_2 will be calculated by the remainder.
- Select the char S conversion path from the **Char S Conversion** drop-down list as **so2**, **h2s**, or **so2/h2s**.
- Specify the mass fraction of sulfur in the char in the **Char S Mass Fraction** field.
- Specify the overall fraction of the char S, by mass, that will be converted to the intermediate species and/or product SO_2 in the **Conversion Fraction** field.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you selected **so2/h2s** from the **Char S Conversion** drop-down list, you must specify the fraction of the converted char S, by mass, that will become SO_2 under **Partition Fractions**. The fraction of char S that will become H_2S will be calculated by the remainder.

The following equations are used to determine the mass fraction of sulfur in the volatiles and char:

$$\dot{m}_{S_{v/c}} = \dot{m}_{v/c} * mf_{S_{v/c}} \tag{21.8}$$

where

$\dot{m}_{S_{v/c}}$ = rate of release of fuel sulfur in kg/s

$\dot{m}_{v/c}$ = rate of release of volatiles (v) or char (c) in kg/s

$mf_{S_{v/c}}$ = mass fraction of sulfur in volatiles or char

Let

TS_{fuel} = total sulfur mass fraction in daf coal (that is, from daf ultimate analysis)

S_{split} = char sulfur as a fraction of total sulfur

F_{vol} = mass fraction of volatiles in daf coal

F_{char} = mass fraction of char in daf coal

Then the following should hold:

$$F_{vol} + F_{char} = 1 \tag{21.9}$$

$$\frac{F_{char} * mf_{S_c}}{TS_{fuel}} = S_{split} \tag{21.10}$$

$$F_{vol} * mf_{S_v} + F_{char} * mf_{S_c} = TS_{fuel} \tag{21.11}$$

$$mf_{S_v} = \left(1 - S_{split}\right) * \frac{TS_{fuel}}{F_{vol}} \tag{21.12}$$

$$mf_{S_c} = S_{split} * \frac{TS_{fuel}}{F_{char}} \tag{21.13}$$

21.2.1.4. Defining the SOx Formation Model Parameters

You can set the SO_x formation model parameters that apply to all of the fuel streams in the **Formation Model Parameters** group box:

- You have the option of including SO_3 as a product, and including SH and SO as intermediates by enabling the **Include SO3 Product** and the **Include SH and SO Intermediates** options, respectively. See Reaction Mechanisms for Sulfur Oxidation in the Theory Guide for further information.

- Specify the method by which O and OH will be calculated. The SO_x routines employ three methods for reduction calculations of SO_x:

  – You can select **equilibrium**, **partial-equilibrium**, or **instantaneous** in the **[O] Model** drop-down list.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– You can select **none**, **partial-equilibrium**, or **instantaneous** in the **[OH] Model** drop-down list.

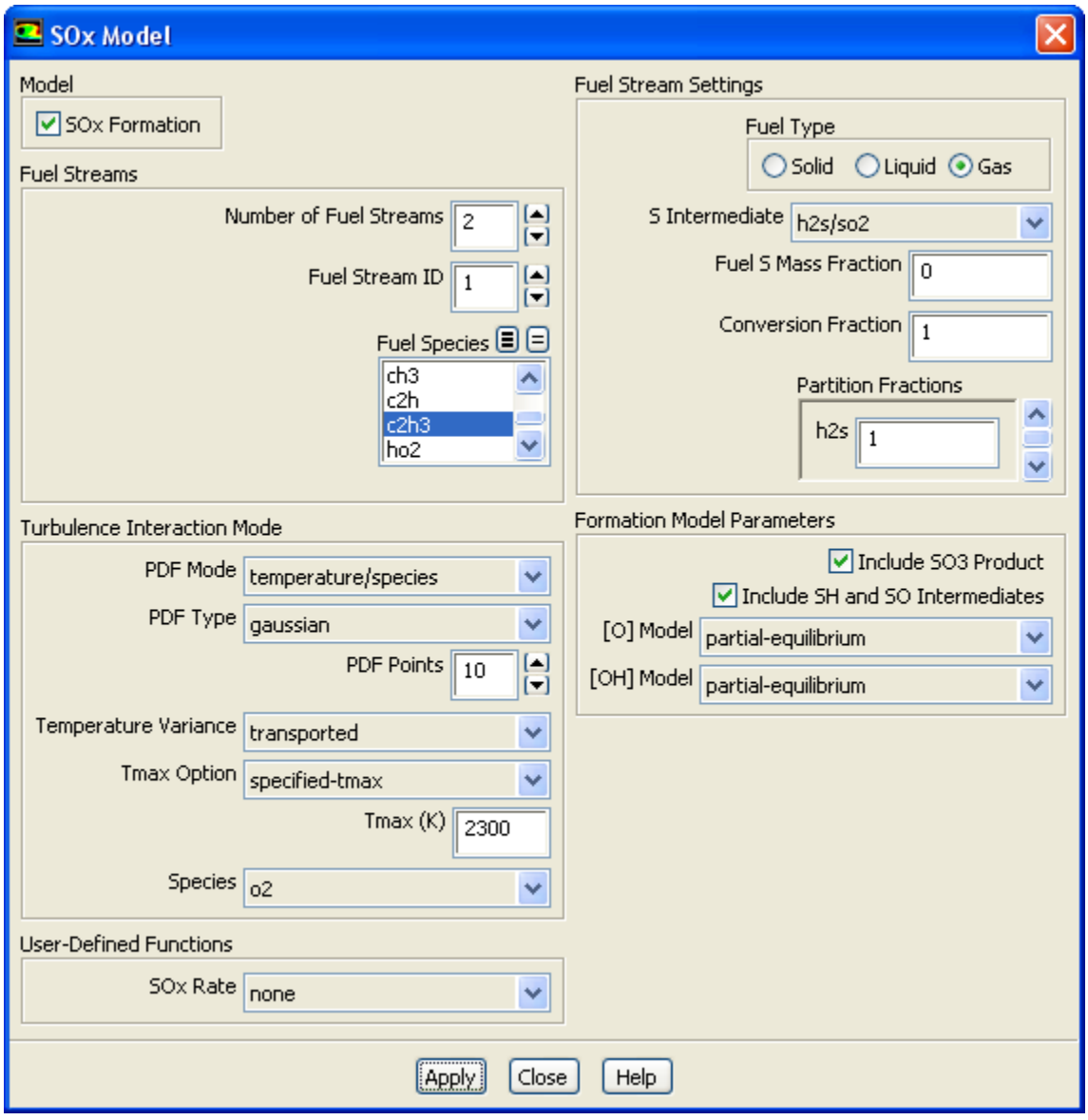
Important

To use the predicted O and/or OH concentration, select **instantaneous** in the **[O] Model** or **[OH] Model** drop-down list.

21.2.1.5. Setting Turbulence Parameters

If you want to take into account turbulent fluctuations when you compute the specified SO_2 formation, define the turbulence parameters in the **Turbulence Interaction Mode** group box.

Figure 21.9: The SOx Model Dialog Box for a Gas Fuel Type with Turbulence



Select one of the options in the **PDF Mode** drop-down list:

- Select **temperature** to take into account fluctuations of temperature.

- Select **temperature/species** to take into account fluctuations of temperature and mass fraction of the species selected in the **Species** drop-down list (which appears when you select this option).
- (non-premixed and partially premixed combustion calculations only) Select **mixture fraction** to take into account fluctuation in the mixture fraction(s).

Important

When modeling the formation of other pollutants along with SO_x, you should compare the selections made in the **PDF Mode** drop-down lists in the **Turbulence Interaction Mode** tab of the NOx Model Dialog Box (p. 2250) and the **Turbulence Interaction Mode** group boxes of the **SOx Model** and **Soot Model** dialog boxes. If **mixture fraction** is selected in any of these dialog boxes, then it must be selected in all of the others as well.

The **mixture fraction** option is available only if you are using either the non-premixed or partially premixed combustion model to model the reacting system. If you use the **mixture fraction** option, the instantaneous temperatures and species concentrations are taken from the PDF look-up table as a function of mixture fraction and enthalpy and the instantaneous SO_x rates are calculated at each cell. The PDF used for convoluting the instantaneous SO_x rates is the same as the one used to compute the mean flow-field properties. For example, for single-mixture fraction models the beta PDF is used, and for two-mixture fraction models, the beta or the double delta PDF can be used. The PDF for mixture fraction is calculated from the values of mean mixture fraction and variance at each cell, and the instantaneous SO_x rates are convoluted with the mixture fraction PDF to yield the mean rates in turbulent flow.

If you selected **temperature** or **temperature/species** for the **PDF Mode**, you should define the following parameters in the **Turbulence Interaction Mode** group box:

PDF Type

allows you to specify the shape of the PDF, which is then integrated to obtain mean rates for the temperature and (if you selected **temperature/species** for the **PDF Mode**) the species. If you select **beta**, the PDF will be modeled using Equation 14.108 in the Theory Guide. If you select **gaussian**, the PDF will be modeled using Equation 14.111 in the Theory Guide.

PDF Points

allows you to specify the number of points used to integrate the beta or Gaussian function in Equation 14.105 or Equation 14.106 in the Theory Guide on a histogram basis. The default value of 10 is a minimum value. It is recommended that you run the calculation with this minimum value to convergence, and then keep increasing the value (for example, 20–25) until the pollutants of interest stop changing. Increasing this value may improve accuracy, but will also increase the computation time.

Temperature Variance

allows you to specify the form of transport equation that is solved to calculate the temperature variance. The default selection is **algebraic**, which is an approximate form of the transport equation (see Equation 14.114 in the Theory Guide). You have the option of selecting **transported** to instead solve Equation 14.113 in the Theory Guide. Though the **transported** form is more exact, it is also more expensive computationally.

Tmax Option

provides various options for determining the maximum limit of the temperature used for the integration of the PDF (see Equation 8.43 in the *Fluent Theory Guide*) to calculate mean rates for turbulent fluctuations of SO_x:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The default selection is **global-tmax**, which sets the limit as the maximum temperature in the flow field.
- You can select **local-tmax** if you would rather obtain cell-based maximum temperature limits by multiplying the local cell mean temperature by the value entered in **Tmax Factor**.
- You can select **specified-tmax** to set the limit for each cell to be the value entered in **Tmax**.
- If you have selected a user-defined function from the **SOx Rate** drop-down menu in the **User-Defined Functions** group box, then you can select **user-defined** so that the limit is specified by a user-defined function. See the Fluent Customization Manual for details about user-defined functions.

Species

only appears if you have selected **temperature/species** for the **PDF Mode**. Your selection in this drop-down menu determines which species' mass fraction is included in the SO_x formation calculations.

Important

Note that the species variance will always be calculated using the algebraic form of the transport equation (Equation 14.114 in the Theory Guide).

21.2.1.6. Specifying a User-Defined Function for the SOx Rate

You can choose to specify a user-defined function for the rate of SO_x production. By default, the rate returned from the user-defined function is added to the rate returned from the standard SO_x production options. You also have the option of replacing ANSYS Fluent's SO_x rate calculations with your own user-defined SO_x rate.

In addition to or instead of using the UDF to specify the SO_x rate, you can use it to specify custom values for the maximum limit (T_{max}) that is used for the integration of the temperature PDF (when temperature is accounted for in the turbulence interaction modeling).

To use a UDF to add a rate to ANSYS Fluent's SO_x rate calculations, you must compile and load the desired function, and then select it from the **SOx Rate** drop-down list in the **User-Defined Functions** group box. After you have selected the UDF, you have the following options:

- You can specify that your custom rate is added to the ANSYS Fluent SO_x rate calculations, by retaining the default selection of **Add to the Fluent Rate** in the **UDF Rate** group box.
- You can replace the ANSYS Fluent SO_x rate calculations with your custom rate, by selecting **Replace Fluent Rate** in the **UDF Rate** group box.
- You can specify custom values for T_{max}, by selecting **user-defined** from the **Tmax Option** drop-down list in the **Turbulence Interaction Mode** group box.

See the Fluent Customization Manual for details about user-defined functions.

21.2.1.7. Defining Boundary Conditions for the SOx Model

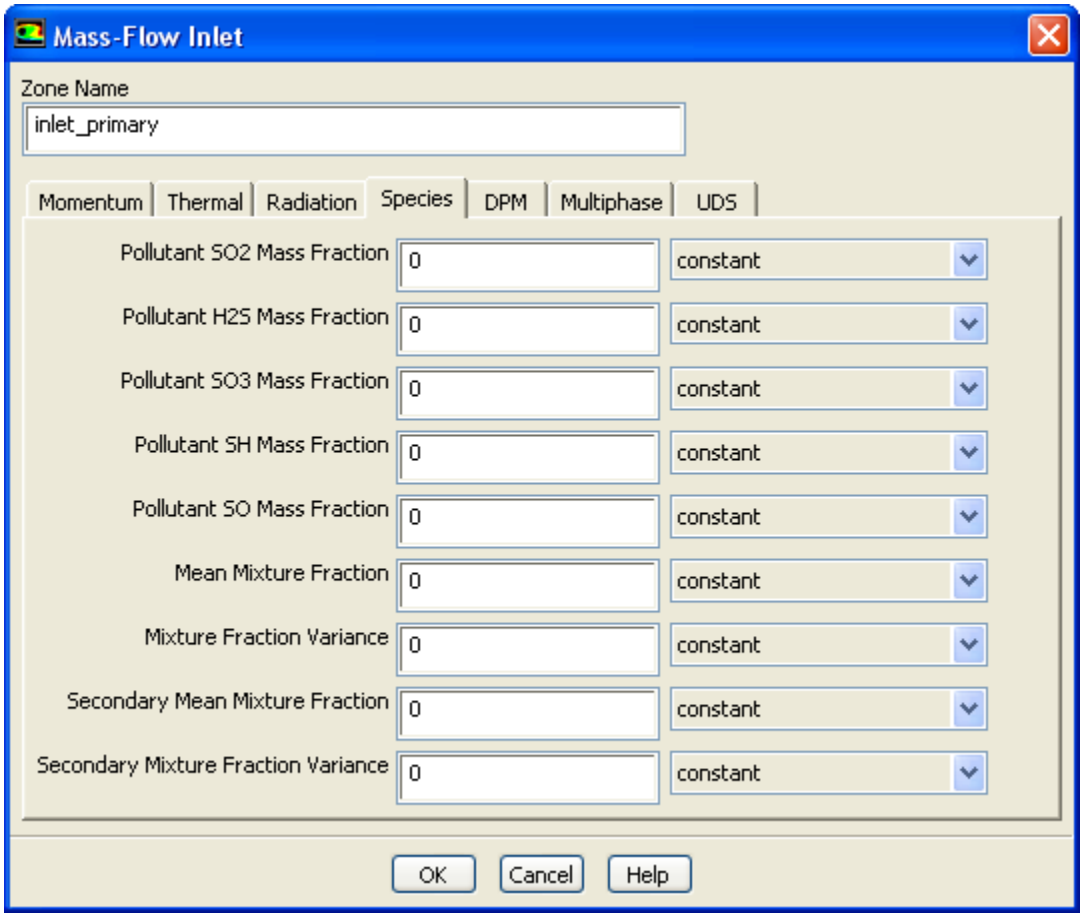
At flow inlet boundaries, you must specify the **Pollutant SO Mass Fraction**, and if necessary, the **Pollutant SH Mass Fraction**, **Pollutant H2S Mass Fraction**, **Pollutant SO3 Mass Fraction**, and **Pollutant SO2 Mass Fraction** in the **Species** tab, as demonstrated in Figure 21.10: The Mass-Flow Inlet Dialog Box and the **Species** Tab (p. 1182).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → Boundary Conditions

You can retain the default inlet values of zero for these quantities or you can enter nonzero numbers as appropriate for your combustion system.

Figure 21.10: The Mass-Flow Inlet Dialog Box and theSpecies Tab



21.2.2. Solution Strategies

To solve for SO_x products:

- (optional) If the discrete phase model (DPM) is turned on (by enabling the **Interaction with Continuous Phase**) to run with the SO_x model, then set the **DPM Iteration Interval** to `0` such that no DPM iterations are performed as the SO_x case is being solved.
- In the Equations Dialog Box (p. 2511) of the Solution Controls Task Page (p. 2508), turn off the solution of all variables except **Pollutant so2** and **Pollutanth2s** (and **Pollutant so3**, **Pollutant sh**, and **Pollutant so**, if applicable).

  Solution → Controls → Equations...
- In the Solution Controls Task Page (p. 2508), set suitable values for the pollutant SO_2 and H_2S (and SO_3, SH, and SO, if applicable) under-relaxation factors. A value of 1.0 is suggested, although lower values may be required for certain problems. That is, if convergence cannot be obtained, try a lower under-relaxation value.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution → **Controls**

- Under **Spatial Discretization** in the Solution Methods Task Page (p. 2504), select the desired scheme for each of the pollutants, SO_2 and H_2S (and SO_3, SH, and SO, if applicable)

Solution → **Methods**

- In the Residual Monitors Dialog Box (p. 2789), decrease the convergence criterion for the pollutants SO_2 and H_2S (and SO_3, SH, and SO, if applicable) to 10^{-6}.

Solution → **Monitors** → **Residuals** **Edit...**

- Perform calculations until convergence (that is, until the SO_2 and H_2S—and SO_3, SH, and SO—pollutant residuals are below 10^{-6}) to ensure that the SO_2 and H_2S (and SO_3, SH, and SO) concentration fields are no longer evolving.

Solution → **Run Calculation**

- Review the mass fractions of pollutants, SO_2 and H_2S (and SO_3, SH, and SO), by generating graphical plots or alphanumeric reports as described in Postprocessing (p. 1183).

21.2.3. Postprocessing

When you compute SO_x formation, the following additional variables will be available for postprocessing. They are contained in the **SOx...** category of the variable selection drop-down list that appears in postprocessing dialog boxes.

- **Mass fraction of pollutant so2**
- **Mass fraction of pollutant h2s**
- **Mass fraction of pollutant so3**
- **Mass fraction of pollutant sh**
- **Mass fraction of pollutant so**
- **Mole fraction of pollutant so2**
- **Mole fraction of pollutant h2s**
- **Mole fraction of pollutant so3**
- **Mole fraction of pollutant sh**
- **Mole fraction of pollutant so**
- **so2 Density**
- **h2s Density**
- **so3 Density**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **sh Density**
- **so Density**
- **Rate of so2**
- **Rate of h2s**
- **Rate of so3**
- **Rate of sh**
- **Rate of so**

21.3. Soot Formation

This section contains information about using the soot formation models in ANSYS Fluent. For information about the theory behind the soot models in ANSYS Fluent, see Soot Formation in the Theory Guide.

21.3.1. Using the Soot Models

21.3.1. Using the Soot Models

When the mass fraction of soot is relatively large (for example, 10%) or if your problem involves the effect of radiation, the soot formation should be computed as part of the main combustion solution and not through postprocessing (as is done for the NO_x and SO_x models). The procedure for setting up and solving a soot formation model is outlined below, and described in detail on the pages that follow. Remember that only the steps that are pertinent to soot modeling are shown here. For information about inputs related to other models that you are using in conjunction with the soot formation model, see the appropriate sections for those models.

1. Set up your combustion problem using ANSYS Fluent as usual. Note the following limitations:
   - None of the soot models are compatible with premixed combustion.
   - The one-step and two-step soot formation models are only available for turbulent flows.
2. Enable the desired soot formation model and set the related parameters, as described in this section.

   Setup → **Models** → **Species** → **Soot** **Edit...**
3. Define the boundary conditions for soot (and nuclei, if you are not using the one-step model) at flow inlets.

   Setup → **Boundary Conditions**
4. In the Solution Controls Task Page (p. 2508), set a suitable value for the soot (and nuclei, if you are not using the one-step model) under-relaxation factor(s). The default value is 0.9. If convergence cannot be obtained with this value, try a lower under-relaxation value.

   Solution → **Controls**
5. Perform calculations until convergence (that is, until the soot / nuclei residual is below 10^{-6}) to ensure that the soot (and nuclei) field is no longer evolving.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution → **Run Calculation**

Note that when **Soot-Radiation Interaction** is enabled in the **Soot Model** dialog box, soot equations (soot, nuclei, and tar species, if available) must be solved together with the combustion solution to obtain correct radiation coupling. Therefore, when **Soot-Radiation Interaction** is enabled, the soot equations are not available for pollutant postprocessing, but will be solved together with the combustion solution. However, if other pollutants are to be solved (while soot/radiation interaction is enabled), those pollutant equations can be postprocessed.

6. Review the mass fraction of soot (and nuclei) by generating graphical plots or alphanumeric reports in the usual way.

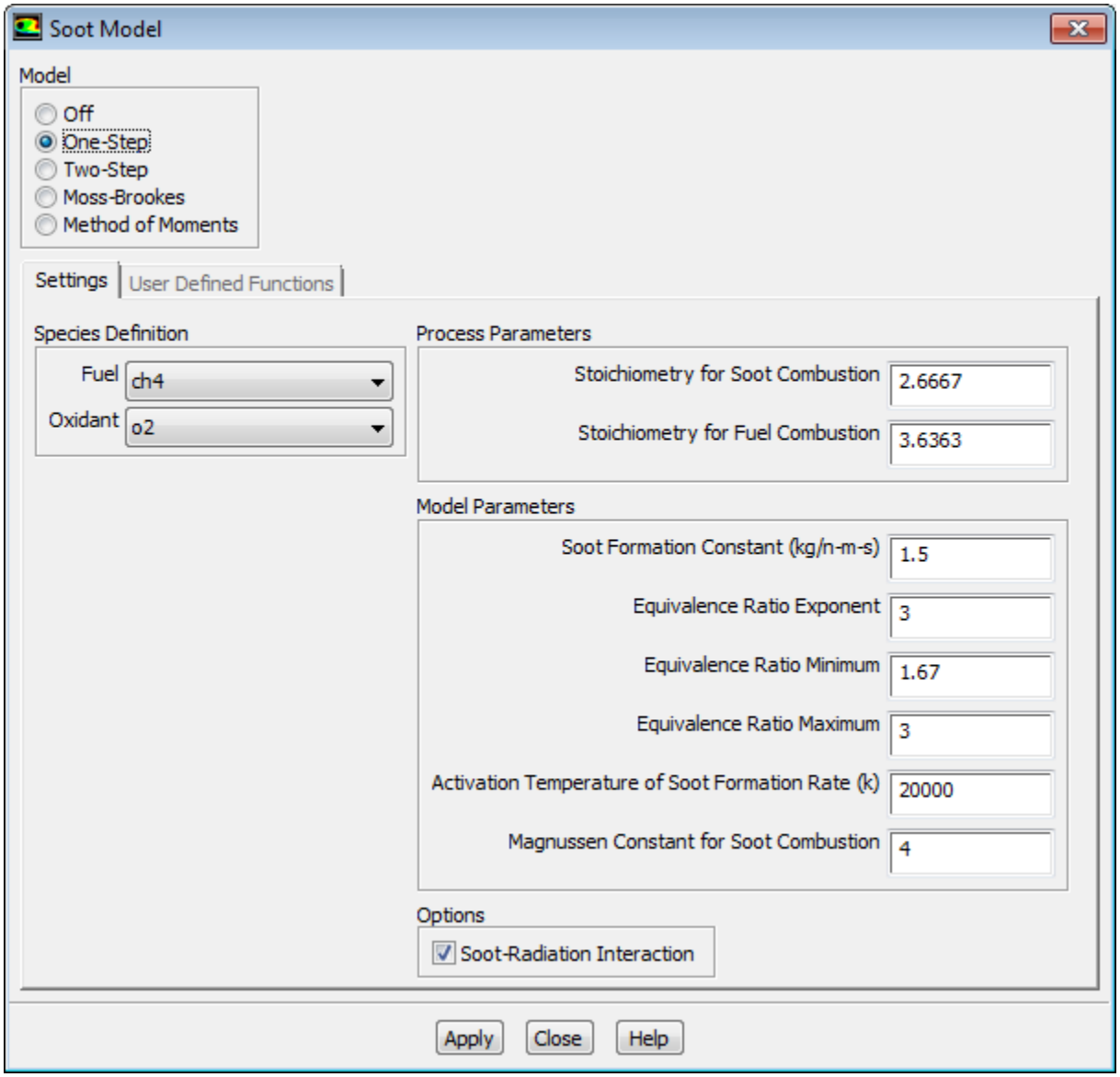
7. Save a new set of case and data files, if desired.

21.3.1.1. Setting Up the One-Step Model

You can enable and set up the one-step soot formation model by using the Soot Model Dialog Box (p. 2262) (Figure 21.11: The Soot Model Dialog Box for the One-Step Model (p. 1186)).

Setup → **Models** → **Species** → **Soot** **Edit...**

Figure 21.11: The Soot Model Dialog Box for the One-Step Model



Under **Model**, select **One-Step**. The dialog box will expand to show the appropriate inputs.

Next, you must tell ANSYS Fluent which chemical species in your model should be used as the fuel and oxidizer. Under **Species Definition**, select the fuel in the **Fuel** drop-down list and the oxidizer in the **Oxidant** drop-down list. If you are using the non-premixed model for the combustion calculation and your fuel stream consists of a mixture of components, you should choose the most appropriate species as the **Fuel** species for the soot formation model. Similarly, the most significant oxidizing component (for example, O_2) should be selected as the **Oxidant**.

If you want to include the effects of soot formation on the radiation absorption coefficient, enable **Soot-Radiation Interaction** in the **Options** group box. For more details, see The Effect of Soot on the Absorption Coefficient in the Theory Guide.

You must next define the **Process Parameters**, by specifying the stoichiometry of the fuel and soot combustion for the one-step model:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stoichiometry for Soot Combustion
is the mass stoichiometry, ν_{soot}, in Equation 14.131 in the Theory Guide, which computes the soot combustion rate. The default value supplied by ANSYS Fluent (2.6667) assumes that the soot is pure carbon and that the oxidizer is O_2.

Stoichiometry for Fuel Combustion
is the mass stoichiometry, ν_{fuel}, in Equation 14.131 in the Theory Guide, which computes the soot combustion rate. The default value supplied by ANSYS Fluent (3.6363) is for combustion of propane ($C_3 H_8$) by oxygen (O_2).

You must then set the **Modeling Parameters** that are used in Equation 14.128, Equation 14.130, and Equation 14.131 in the Theory Guide:

Soot Formation Constant
is the parameter C_s in Equation 14.128 in the Theory Guide.

Equivalence Ratio Exponent
is the exponent r in Equation 14.128 in the Theory Guide.

Equivalence Ratio Minimum
and **Equivalence Ratio Maximum** are the minimum and maximum values of the fuel equivalence ratio ϕ in Equation 14.128 in the Theory Guide. This equation will be solved only if **Equivalence Ratio Minimum** $<\phi<$ **Equivalence Ratio Maximum**; if ϕ is outside of this range, there is no soot formation.

Activation Temperature of Soot Formation Rate
is the term E / R in Equation 14.128 in the Theory Guide.

Magnussen Constant for Soot Combustion
is the constant A used in the rate expressions governing the soot combustion rate (Equation 14.130 and Equation 14.131 in the Theory Guide).

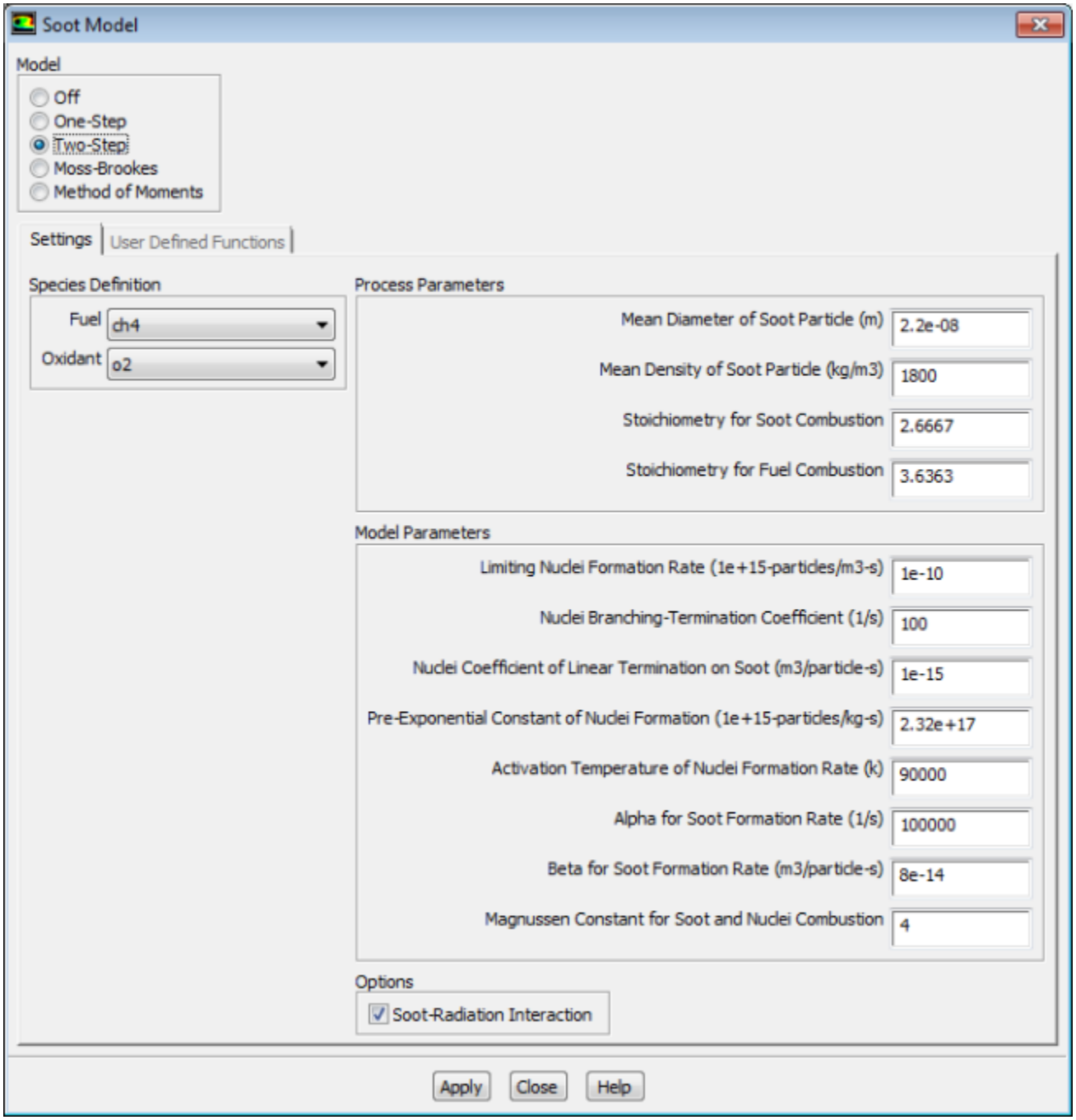
Note that the default values for these parameters are for propane fuel [21] (p. 2888), [126] (p. 2893), and are considered to be valid for a wide range of hydrocarbon fuels.

21.3.1.2. Setting Up the Two-Step Model

You can enable and set up the two-step soot formation model by using the Soot Model Dialog Box (p. 2262) (Figure 21.12: The Soot Model Dialog Box for the Two-Step Model (p. 1188)).

Setup → Models → Species → Soot → Edit...

Figure 21.12: The Soot Model Dialog Box for the Two-Step Model



Under **Model**, select **Two-Step**. The dialog box will expand to show the appropriate inputs.

Important

Note that the two-step Tesner model should only be used when the eddy-dissipation model is used to define the turbulence-chemistry interaction.

Next, you must tell ANSYS Fluent which chemical species in your model should be used as the fuel and oxidizer. Under **Species Definition**, select the fuel in the **Fuel** drop-down list and the oxidizer in the **Oxidant** drop-down list. If you are using the non-premixed model for the combustion calculation and your fuel stream consists of a mixture of components, you should choose the most appropriate species

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

as the **Fuel** species for the soot formation model. Similarly, the most significant oxidizing component (for example, O_2) should be selected as the **Oxidant**.

If you want to include the effects of soot formation on the radiation absorption coefficient, enable **Soot-Radiation Interaction** in the **Options** group box. For more details, see The Effect of Soot on the Absorption Coefficient in the Theory Guide.

You must next define the **Process Parameters**, by specifying the stoichiometry of the fuel and soot combustion, as well as the average size and density of the soot particles, for the two-step model:

Mean Diameter of Soot Particle

and **Mean Density of Soot Particle** are the assumed average diameter and average density of the soot particles in the combustion system, used to compute the soot particle mass, m_p, in Equation 14.134 in the Theory Guide for the two-step model. Note that the default values for soot density and diameter are taken from [64] (p. 2890).

Stoichiometry for Soot Combustion

is the mass stoichiometry, ν_{soot}, in Equation 14.131 in the Theory Guide, which computes the soot combustion rate. The default value supplied by ANSYS Fluent (2.6667) assumes that the soot is pure carbon and that the oxidizer is O_2.

Stoichiometry for Fuel Combustion

is the mass stoichiometry, ν_{fuel}, in Equation 14.131 in the Theory Guide, which computes the soot combustion rate. The default value supplied by ANSYS Fluent (3.6363) is for combustion of propane ($C_3 H_8$) by oxygen (O_2).

You must then set the **Modeling Parameters** that are used in Equation 14.130, Equation 14.131, Equation 14.134, Equation 14.136, and Equation 14.137 in the Theory Guide:

Limiting Nuclei Formation Rate

is the limiting value of the kinetic nuclei formation rate, η_0 in Equation 14.137 in the Theory Guide. Below this limiting value, the branching and termination term, $(f-g)$ in Equation 14.136 in the Theory Guide, is not included.

Nuclei Branching-Termination Coefficient

is the term $\left(f-g\right)$ in Equation 14.136 in the Theory Guide.

Nuclei Coefficient of Linear Termination on Soot

is the term g_0 in Equation 14.136 in the Theory Guide.

Pre-Exponential Constant of Nuclei Formation

is the pre-exponential term a_0 in the kinetic nuclei formation term, Equation 14.137 in the Theory Guide.

Activation Temperature of Nuclei Formation Rate

is the term E/R in the kinetic nuclei formation term, Equation 14.137 in the Theory Guide.

Alpha for Soot Formation Rate

is α, the constant in the soot formation rate equation, Equation 14.134 in the Theory Guide.

Beta for Soot Formation Rate

is β, the constant in the soot formation rate equation, Equation 14.134 in the Theory Guide.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Magnussen Constant for Soot and Nuclei Combustion
is the constant A used in the rate expressions governing the soot combustion rate (Equation 14.130 and Equation 14.131 in the Theory Guide).

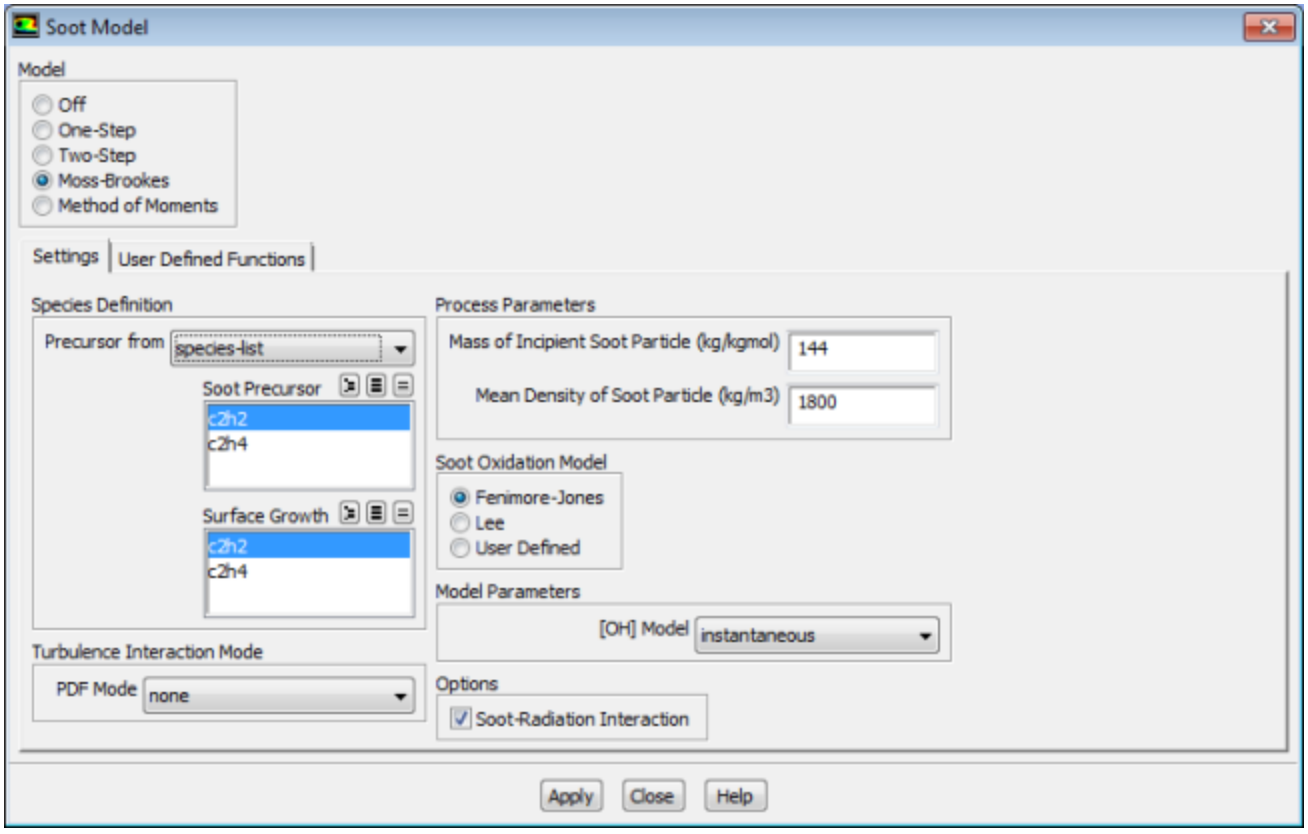
The default values for the two-step model are the same as in Magnussen and Hjertager [64] (p. 2890) (for an acetylene flame), except for a_0, which is assumed to have the original value from Tesner et al. [122] (p. 2893). If your model involves propane fuel rather that acetylene, it is recommended that you change the value of α to 3.5×10^8 [5] (p. 2887). For best results, you should modify both of these parameters, using empirically determined inputs for your specific combustion system.

21.3.1.3. Setting Up the Moss-Brookes Model and the Hall Extension

You can enable and set up the Moss-Brookes and Moss-Brookes-Hall soot formation models by using the Soot Model Dialog Box (p. 2262) (Figure 21.13: The Soot Model Dialog Box for the Moss-Brookes Model (p. 1190)).

Setup → Models → Species → Soot Edit...

Figure 21.13: The Soot Model Dialog Box for the Moss-Brookes Model



Under **Model**, select **Moss-Brookes** or **Moss-Brookes-Hall**. The dialog box will expand to show the appropriate inputs. Note the following about these models:

- The **Moss-Brookes** model was originally proposed for soot prediction in methane flames. However, it can be equally applicable to higher hydrocarbon species by appropriately modifying the soot precursor and participating surface growth species.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The **Moss-Brookes-Hall** model is applicable for higher hydrocarbon fuels (for example, kerosene) and will only be available when C_2H_2, C_6H_6, C_6H_5, and H_2 are present in the gas phase species list.

You must next define the precursor species in the **Species Definition** group box. When suitable precursor species are present in the species list, you can select **species-list** from the **Precursor from** drop-down list, and then select the **Soot Precursor** species and the **Surface Growth** species from the selection lists. Note that for the **Moss-Brookes** model, you can select acetylene (**c2h2**), ethylene (**c2h4**), and/or benzene (**c6h6**) for the **Soot Precursor**; if neither are present or if you would like to specify a different precursor correlation, then curve fitting will be used to determine the precursor and surface growth species mass fractions (see Species Definition for the Moss-Brookes Model with a User-Defined Precursor Correlation (p. 1194) for further details regarding curve fitting).

Next, specify how turbulent fluctuations will be accounted for in the soot formation calculations, by defining the turbulence parameters in the **Turbulence Interaction Mode** group box.

Select one of the options in the **PDF Mode** drop-down list:

- Select **none** to ignore turbulence and use laminar soot rate calculations.
- Select **temperature** to take into account fluctuations of temperature.
- Select **temperature/species** to take into account fluctuations of temperature and mass fraction of the species selected in the **Species** drop-down list (which appears when you select this option).
- (non-premixed and partially premixed combustion calculations only) Select **mixture fraction** to take into account fluctuation in the mixture fraction(s). This is the recommended approach, as it generally yields the best results and accuracy.

Important

When modeling the formation of other pollutants along with soot, you should compare the selections made in the **PDF Mode** drop-down lists in the **Turbulence Interaction Mode** tab of the NOx Model Dialog Box (p. 2250) and the **Turbulence Interaction Mode** group boxes of the **SOx Model** and **Soot Model** dialog boxes. If **mixture fraction** is selected in any of these dialog boxes, then it must be selected in all of the others as well.

The **mixture fraction** option is available only if you are using either the non-premixed or partially premixed combustion model to model the reacting system. If you use the **mixture fraction** option, the instantaneous temperatures and species concentrations are taken from the PDF look-up table as a function of mixture fraction and enthalpy and the instantaneous soot production rates are calculated at each cell. The PDF used for convoluting the instantaneous soot rates is the same as the one used to compute the mean flow-field properties. For example, for single-mixture fraction models the beta PDF is used, and for two-mixture fraction models, the beta or the double delta PDF can be used. The PDF in terms of mixture fraction is calculated from the values of mean mixture fraction and variance at each cell, and the instantaneous soot rates are convoluted with the mixture fraction PDF to yield the mean rates in turbulent flow.

Note

When `mixture fraction` is selected as **PDF mode**, ANSYS Fluent calculates and tabulates the soot rates apriori for faster computations of soot rates. The soot rates are tabulated at the first soot iteration and a message

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Performing Pollutant PDF integrations...
```

is displayed in the Fluent console.

If you selected **temperature** or **temperature/species** for the **PDF Mode**, you should define the following parameters in the **Turbulence Interaction Mode** group box:

PDF Type
allows you to specify the shape of the PDF, which is then integrated to obtain mean rates for the temperature and (if you selected **temperature/species** for the **PDF Mode**) the species. If you select **beta**, the PDF will be modeled using Equation 14.108 in the Theory Guide. If you select **gaussian**, the PDF will be modeled using Equation 14.111 in the Theory Guide.

PDF Points
allows you to specify the number of points used to integrate the beta or Gaussian function in Equation 14.105 or Equation 14.106 in the Theory Guide on a histogram basis. The default value of 10 is a minimum value. It is recommended that you run the calculation with this minimum value to convergence, and then keep increasing the value (for example, 20–25) until the pollutants of interest stop changing. Increasing this value may improve accuracy, but will also increase the computation time.

Temperature Variance
allows you to specify the form of transport equation that is solved to calculate the temperature variance. The default selection is **algebraic**, which is an approximate form of the transport equation (see Equation 14.114 of the Theory Guide). You have the option of selecting **transported** to instead solve Equation 14.113 in the Theory Guide. Though the **transported** form is more exact, it is also more expensive computationally.

Tmax Option
provides various options for determining the maximum limit of the temperature used for the integration of the PDF (see Equation 8.43 in the *Fluent Theory Guide*) to calculate mean rates for turbulent fluctuations of soot. The default selection is **global-tmax**, which sets the limit as the maximum temperature in the flow field. You can select **local-tmax** if you would rather obtain cell-based maximum temperature limits by multiplying the local cell mean temperature by the value entered in **Tmax Factor**. You can select **specified-tmax** to set the limit for each cell to be the value entered in **Tmax**. Finally, if you have compiled a user-defined function for the soot rate and loaded the library into ANSYS Fluent, then you can select **user-defined** so that the limit is specified by a user-defined function.

Species
only appears if you have selected **temperature/species** for the **PDF Mode**. Your selection in this drop-down menu determines which species' mass fraction is included in the soot formation calculations.

Important

Note that the species variance will always be calculated using the algebraic form of the transport equation (Equation 14.114 in the Theory Guide).

Under **Process Parameters**, you must enter information about the mass and mean density of the soot particles:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mass of Incipient Soot Particles
is M_P in Equation 14.142 and Equation 14.144 in the Theory Guide. Note that this value was assumed to be 144 kg/kmol (12 carbon atoms) in the work of Brookes and Moss, whereas the Hall extension model assumed it to be 1200 kg/kmol (100 carbon atoms).

Mean Density of Soot Particle
is ρ_{soot} in Equation 14.141 in the Theory Guide and ρ in Equation 14.144 in the Theory Guide. Note that this value was assumed to be 1800 kg/m^3 in the work of Brookes and Moss [18] (p. 2887), whereas Hall et al. [40] (p. 2889) assumed it to be 2000 kg/m^3.

Next, you must select the **Soot Oxidation Model**. Your choices include the **Fenimore-Jones** model, as originally used in Brookes and Moss' work, or the **Lee** extended model. The **Lee** model will model soot oxidation due to hydroxyl radicals as in the **Fenimore-Jones** model, as well as the oxidation due to molecular oxygen.

You must then set the **Modeling Parameters**:

[OH] Model
allows you to specify the method by which the OH radical concentration is calculated. The recommended selection from the drop-down list is **instantaneous**, although this option is only available when OH is available in the species list and is calculated by the combustion model. The other option is the **partial-equilibrium** model, which necessitates the availability of $[\mathrm{O}]$ atom concentration within the field.

[O] Model
must be defined when you have selected **partial-equilibrium** for the **[OH] Model**, and specifies the method by which the O radical concentration is calculated. The options include **equilibrium**, **partial-equilibrium**, and **instantaneous**.

Note that in ANSYS Fluent, the oxidation rate scaling parameter (C_{oxid} in Equation 14.142 in the Theory Guide) is set to unity. If you would like to change the value of this parameter, you can use the `define/models/soot-parameters` text command. A lower value will reduce the amount of soot oxidation.

If you want to include the effects of soot formation on the radiation absorption coefficient, enable **Soot-Radiation Interaction** in the **Options** group box. For more details, see The Effect of Soot on the Absorption Coefficient in the Theory Guide.

21.3.1.3.1. Specifying a User-Defined Function for the Soot Oxidation Rate

You can choose to specify a user-defined function for the soot oxidation rate. The compiled UDF hook is available from the **User Defined Oxidation Rate** drop-down list of the **Soot Model** dialog box. See `DEFINE_SOOT_OXIDATION_RATE` in the *Fluent Customization Manual* for details about user-defined functions.

21.3.1.3.2. Specifying a User-Defined Function for the Soot Precursor Concentration

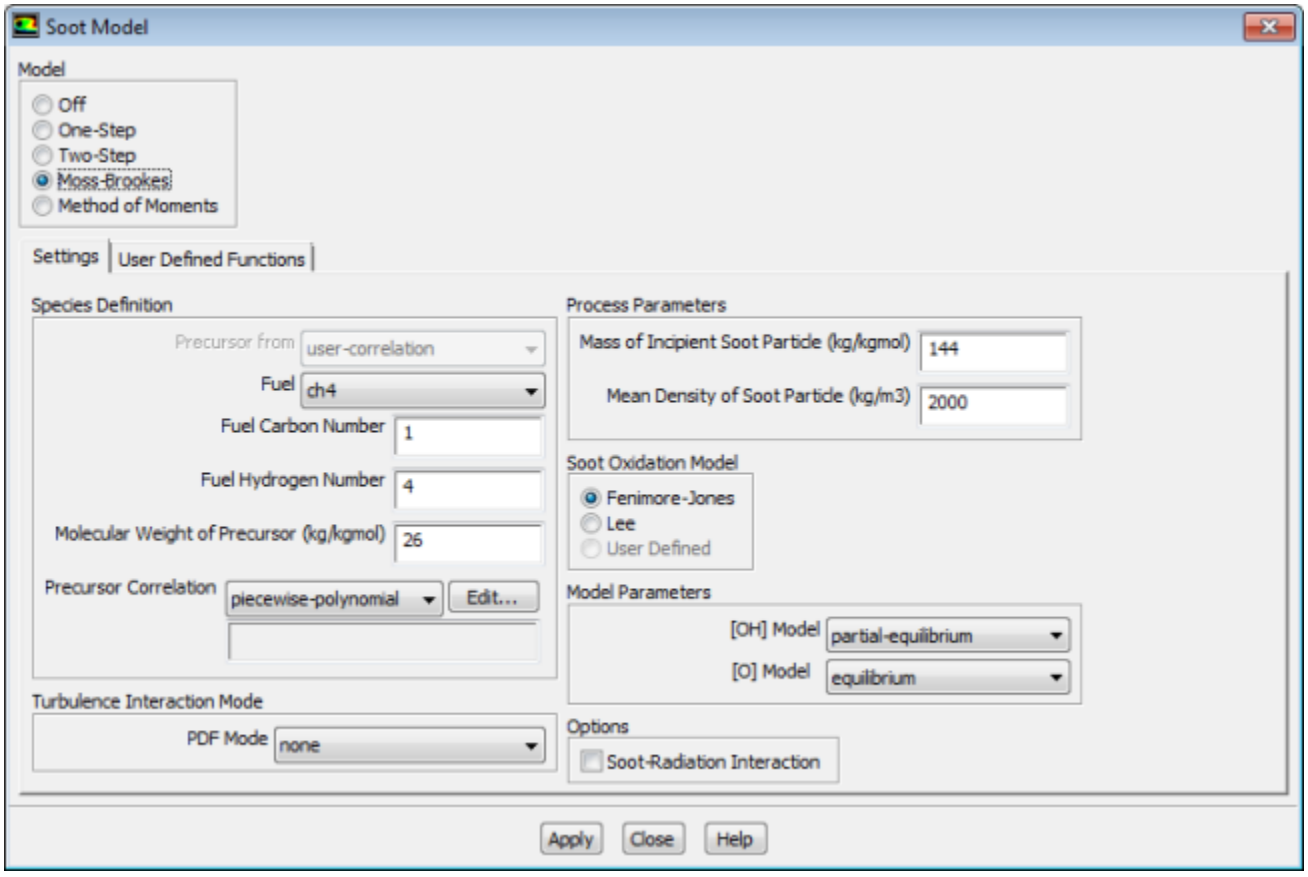
You can choose to specify a user-defined function for the soot oxidation rate. The compiled UDF hook is available from the **User Defined Precursor** drop-down list in the **Species Definition** group box of the **Soot Model** dialog box. See `DEFINE_SOOT_PRECURSOR` in the *Fluent Customization Manual* for details about user-defined functions.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

21.3.1.3.3. Species Definition for the Moss-Brookes Model with a User-Defined Precursor Correlation

ANSYS Fluent accepts the following as possible precursor species for the Moss-Brookes model: C_2H_2, C_6H_6, and C_2H_4. If none of these species are present in the species list (as is often the case when using the eddy-dissipation model) or if you would prefer to specify a different precursor correlation, your setup for the Moss-Brookes model will be different than noted previously. Under such circumstances, you should select **user-correlation** from the **Precursor from** drop-down list in the **Species Definition** group box (note that this is only option possible when the appropriate species are not present). The Soot Model Dialog Box (p. 2262) will then be as shown in Figure 21.14: The Soot Model Dialog Box for the Moss-Brookes Model with a User-Defined Precursor Correlation (p. 1194). The parameters you set in the **Species Definition** group box allow ANSYS Fluent to calculate a mixture fraction based on the mass fractions of the oxidant and the carbon/hydrogen contributed by a designated fuel species. The precursor species mass fraction will then be computed as a function (which you will also define) of this mixture fraction.

Figure 21.14: The Soot Model Dialog Box for the Moss-Brookes Model with a User-Defined Precursor Correlation



In the **Species Definition** group box, you will first select a **Fuel** species and enter the related **Fuel Carbon Number** and **Fuel Hydrogen Number** for use in the mixture fraction calculation. Next, enter the **Molecular Weight of Precursor** (the default value is for acetylene). Then make a selection in the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Precursor Correlation drop-down list to indicate how the precursor mass fraction will be related to the mixture fraction. A **piecewise-polynomial** profile is defined by default.

Important

Note that the default values for the **piecewise-polynomial** profile are only valid for a methane diffusion flame simulation, in which both the air and fuel initial temperatures are set to 290 K, and acetylene is assumed as the soot precursor.

If you decide not to use the default values for **Precursor Correlation**, you must define the correlation between the precursor mass fraction and the mixture fraction. This correlation should be based on a laminar flamelet profile that you have generated: if possible, you should generate the profile using the **Species Model** dialog box (as described in Setting Up the Steady and Unsteady Diffusion Flamelet Models (p. 1036)), being sure to select **Steady Flamelet** in the **Chemistry** tab; if there is no mechanism, you can generate the profile using the equilibrium chemistry model (see Setting Up the Equilibrium Chemistry Model (p. 1031) for details); you may also use a third-party software package of your choosing. You should then apply a curve-fitting technique to your generated profile, to obtain either a constant value or a piecewise-polynomial function.

In a piecewise-polynomial function, the laminar flamelet profile is divided into a number of mixture fraction ranges. In each range, the precursor species mass fraction Y_{prec} is defined using the following equation:

$$Y_{prec} = \sum_{i=1}^{i=NC} C_i f^{i-1} \tag{21.14}$$

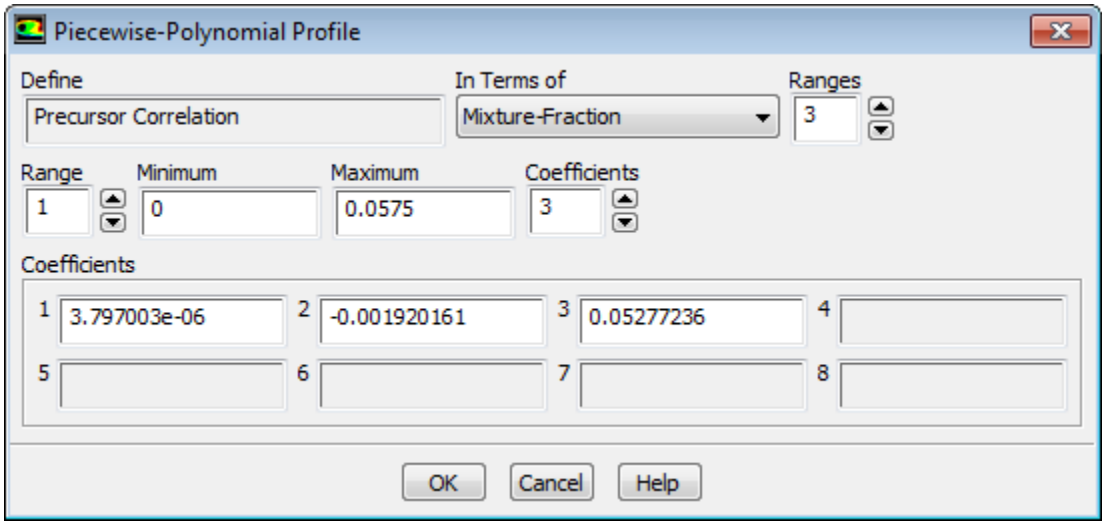
where NC is the number of coefficients C, and f is the mixture fraction. The following piecewise-polynomial function corresponds to the default settings in ANSYS Fluent:

$$Y_{prec} = \begin{cases} \text{for } 0 \le f < 0.0575\text{:} \\ 3.797003\times10^{-6} - 1.920161\times10^{-3} f \\ +5.277237\times10^{-2} f^2 \\ \text{for } 0.0575 \le f < 0.128\text{:} \\ 1.051312 - 71.34743 f + 1964.038 f^2 - 28182.59 f^3 \\ +223543.4 f^4 - 932192 f^5 + 1599627 f^6 \\ \text{for } 0.128 \le f < 1\text{:} \\ 7.988928\times10^{-3} - 8.440912\times10^{-3} f \\ +4.273195\times10^{-4} f^2 \end{cases} \tag{21.15}$$

To define a piecewise-polynomial profile to relate the precursor mass fraction to the mixture fraction, select **piecewise-polynomial** from the **Precursor Correlation** drop-down list and click the **Edit...** button. The Piecewise-Polynomial Profile Dialog Box (p. 2319) (Figure 21.15: The Piecewise-Polynomial Profile Dialog Box (p. 1196)) will open.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 21.15: The Piecewise-Polynomial Profile Dialog Box



Then perform the following steps in the **Piecewise-Polynomial Profile** dialog box:

1. Enter the number of **Ranges**. For the example shown in Equation 21.15 (p. 1195), three ranges of mixture fraction are defined, which is the maximum allowed.

2. For the first range (**Range** = 1), enter the **Minimum** and **Maximum** mixture fraction values, and the number of **Coefficients**. (Up to eight coefficients are available.) The number of coefficients defines the order of the polynomial. An input of 1 defines a polynomial of order 0, and the mass fraction will be constant and equal to the single coefficient. An input of 2 defines a polynomial of order 1, and the mass fraction will vary linearly with mixture fraction, and so on.

3. Define the values for the coefficients in the **Coefficients** group box. The dialog box in Figure 21.15: The Piecewise-Polynomial Profile Dialog Box (p. 1196) shows the inputs for the first range of Equation 21.15 (p. 1195).

4. Increase the value of **Range** and enter the **Minimum** and **Maximum** mixture fractions, number of **Coefficients**, and the values for the **Coefficients** for the next range. Repeat if there is a third range.

Important

Note when defining the ranges, you must start with the lowest mixture fraction range, and then proceed in order to the highest range. The solver will *not* sort them for you.

To define a constant profile to relate the precursor mass fraction to the mixture fraction, select **constant** from the **Precursor Correlation** drop-down list and enter a value in the accompanying text entry box.

21.3.1.4. Setting Up the Method of Moments Soot Model

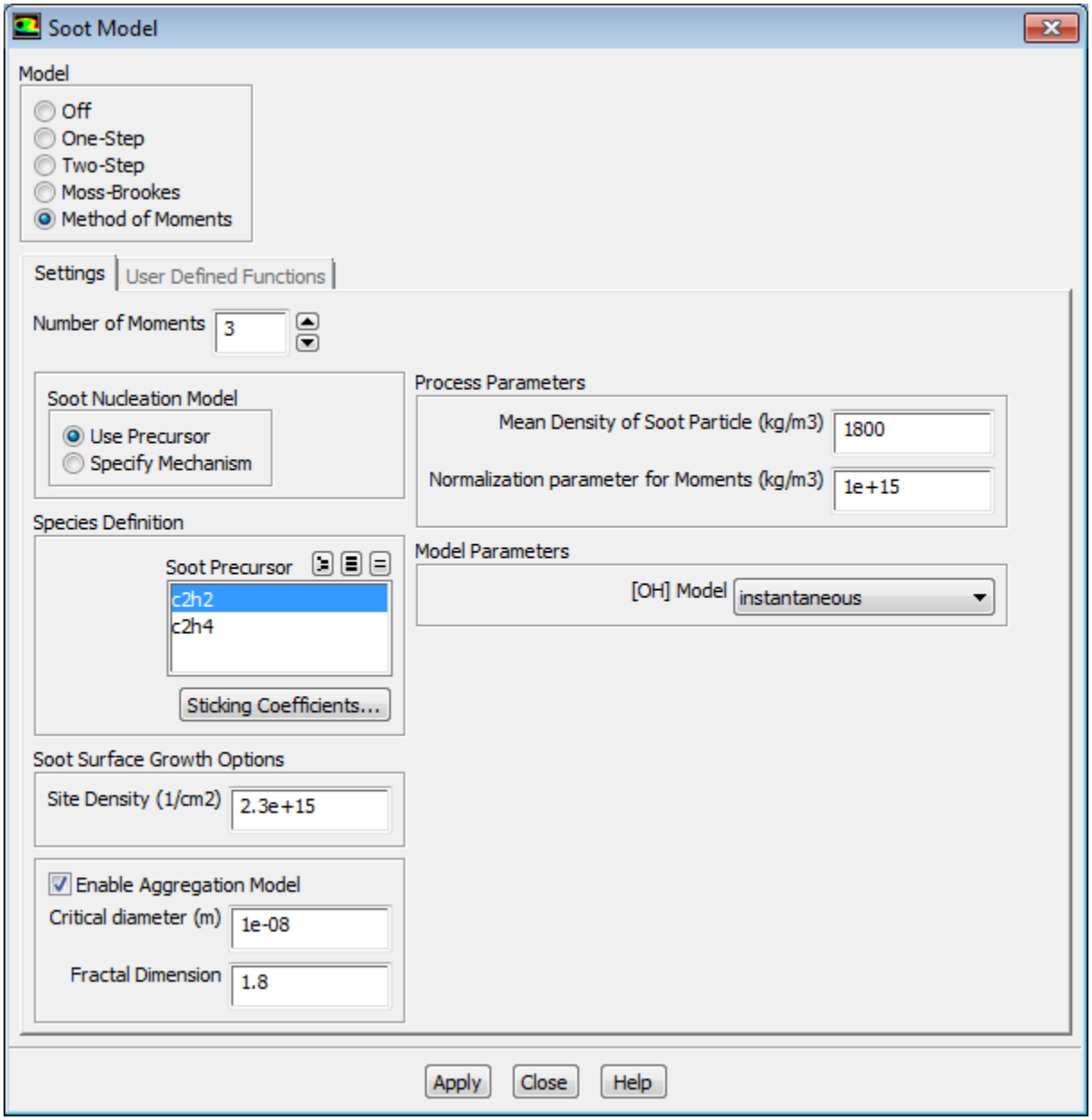
You can enable and set up the Method of Moments model by using the **Soot Model** dialog box (Figure 21.16: The Soot Model Dialog Box for the Method of Moments Model (p. 1197)).

Setup → Models → Species → Soot → Edit...

1. In the **Soot Model** dialog box under **Model**, select **Method of Moments**. The dialog box will expand to show the appropriate inputs.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 21.16: The Soot Model Dialog Box for the Method of Moments Model



2. Specify **Number Of Moments**.

   ANSYS Fluent will solve an equal number of moment transport equations. The default value of 3 moments works reasonably well for a wide range of applications. However, you can specify up to 6 moments to achieve higher accuracy.

3. From the **Soot Nucleation Model** group box, select one of the following options:

   - **Use Precursor** (default): Enables you to define the precursor species for your soot prediction simulation.
   - **Specify Mechanism**: Enables you to provide the nucleation rate and specify nucleation as a reaction between soot precursors. Using this option, you can achieve more accurate results where the nucleation rate data is available.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

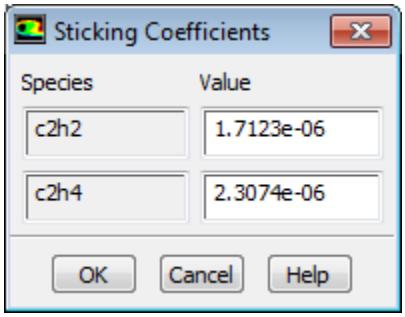
4. (if **Use Precursor** is selected for the soot nucleation model) Under the **Species Definition** group box, define precursor species.

   a. In the **Soot Precursor** group list (which is populated with all precursors available in your analysis), select one or more precursors.

      The soot precursor species must be a hydrocarbon species. ANSYS Fluent uses kinetic theory to calculate the nucleation rates considering coagulation between precursor species.

   b. Click **Sticking Coefficients...** to specify the sticking coefficient for each precursor in the **Sticking Coefficients** dialog box that will open.

      Figure 21.17: Sticking Coefficients for Soot Precursors



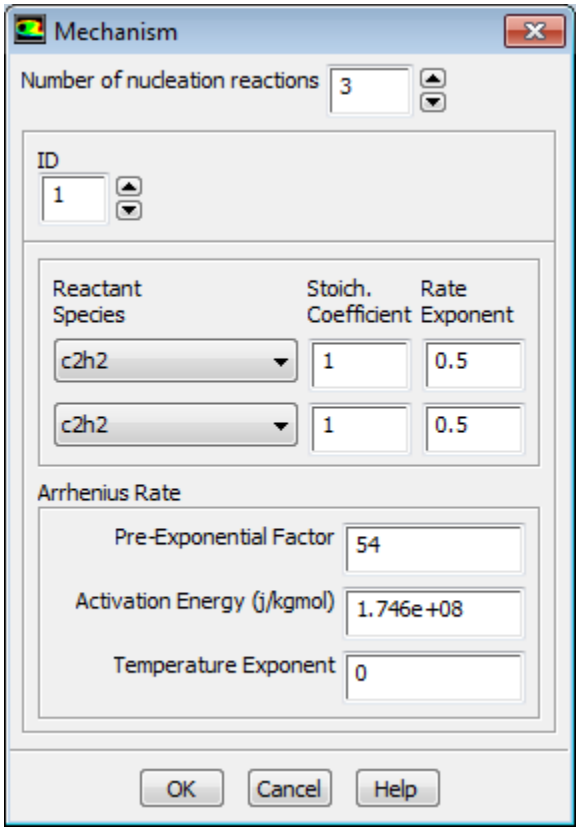
      You must specify the sticking coefficient for each precursor because the nucleation rates calculated based on kinetic theory of collision of two soot precursors are generally very large, causing numerical errors.

> **Important**
>
> The default value for a sticking coefficient is calculated base on precursor molecular weight. In general, you can use smaller values for sticking coefficients for smaller precursors. For guidelines on specifying the sticking coefficients for precursor species, see Nucleation in the *Fluent Theory Guide*.

5. (if **Specify Mechanism** is selected for the soot nucleation model) Click **Set Nucleation Mechanism...** and enter the reaction rates inputs in the **Mechanism** dialog box that will open.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 21.18: Settings for the Nucleation Mechanism



a. Specify **Number of nucleation reactions**.

The nucleation reactions can be single or multi-step.

b. For each reaction, specify the following:

- Reactant species and corresponding stoichiometric coefficient and rate exponent
- The **Arhenius Rate** parameters (**Pre-Exponential Factor**, **Activation Energy**, and **Temperature Exponent**)

Note

The inputs in the **Mechanism** dialog box are required only for reactant species as the main product of specified nucleation reaction is always assumed to be a soot nuclei, and all other products are not considered in calculations of soot nucleation rates.

---

6. If you want to account for soot aggregate formation, select **Enable Aggregation Model**

In addition to the moment transport equations, ANSYS Fluent will solve an equal number of the particle moment transport equations.

Once you have enabled the aggregation model, you can specify the following parameters:

Critical diameter: A value of the soot particle diameter above which the coagulation regime is switched from coalescent coagulation to aggregate coagulation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fractal Dimension: A parameter that describes the fractal structure of the aggregate. Values between 1.7 and 2.0 work reasonably good for the aggregate coagulation.

Note

When the **Enable Aggregation Model** option is enabled, the Method of Moments model tends to over-predict the production of soot. This is due, in part, to the linear interpolation of the particle moments. This aspect of the over-prediction can be overcome by employing quadratic interpolation of the particle moments. Another reason for the over-prediction of soot is due to how ANSYS Fluent currently ignores the depletion of acetylene during the soot calculations. Acetylene is consumed during the surface growth of soot, and since ANSYS Fluent generates the flamelets before the solution, the flamelets have no knowledge of the soot process, thereby producing a higher surface growth of soot, and eventually higher soot yields.

For information about the aggregation model, see Soot Aggregation in the *Fluent Theory Guide*.

7. Under **Soot Surface Growth Options**, specify **Soot Density**.

   The growth of soot due to surface reactions and depletion due to oxidation reactions are modeled using HACA mechanism (see Surface Growth and Oxidation in the separate Theory Guide). For reaction rates and reaction steps, ANSYS Fluent uses default values and requires no user inputs except for the soot site density.

   Note

   You can specify and/or edit the reaction rates of surface reactions using the `define/models/soot-parameters` text command.

   Important

   The soot site density is always provided in units of number of sites per cm^2 irrespective of the unit system selected in ANSYS Fluent.

   ---

8. Under **Process Parameters**, specify **Mean Density of Soot Particle** and **Normalization Parameter for Moments**.

   The normalization parameter is used to minimize numerical errors when solving the normalized soot moment transport equations. The default value for this parameter is sufficient, and you may not need to change it.

9. Specify the **Modeling Parameters**:

   - **[OH] Model**: Specifies the method by which the [OH] radical concentration is calculated. For **OH Model**, the following options are available:

     – `instantaneous` (recommended where possible)

       This option is only available when OH is present in the species list and participate in the combustion simulation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - `partial-equilibrium`

    This option necessitates the availability of [O] atom concentration within the field.

- **[O] Model** (available only if `instantaneous` is not selected for **OH Model**): Specifies the method by which the [O] radical concentration is calculated. The following options are available:

  - `equilibrium`
  - `partial-equilibrium`
  - `instantaneous`

21.3.1.5. Defining Boundary Conditions for the Soot Model

At flow inlet boundaries, you must specify the **Soot Mass Fraction** and (when not using the one-step model) the **Nuclei** mass concentration. These correspond to Y_{soot} in Equation 14.126 and Equation 14.140 and b^*_{nuc} in Equation 14.132 and Equation 14.140 in the Theory Guide, respectively.

Setup → **Boundary Conditions**

You can retain the default inlet values of zero for both quantities or you can enter nonzero numbers as appropriate for your combustion system.

21.3.1.6. Reporting Soot Quantities

ANSYS Fluent provides additional reporting options when your model includes soot formation. You can generate graphical plots or alphanumeric reports of the following items:

- **Mass fraction of Soot**
- **Mole fraction of Soot**
- **Soot Density**
- **Soot Volume fraction**
- **Rate of Soot**
- **Normalized Concentration of Nuclei** (unavailable for one-step model)
- **Rate of Nuclei** (unavailable for one-step model)
- **Rate of Soot Mass Nucleation** (Moss-Brookes and Moss-Brookes-Hall models only)
- **Rate of Surface Growth** (Moss-Brookes and Moss-Brookes-Hall models only)
- **Rate of Oxidation** (Moss-Brookes and Moss-Brookes-Hall models only)
- **Rate of Nucleation** (Moss-Brookes and Moss-Brookes-Hall models only)
- **Rate of Coagulation** (Moss-Brookes and Moss-Brookes-Hall models only)
- **Soot Surface Area** (Method of Moments model only)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Soot Mean Diameter** (Method of Moments model only)
- **Normalized soot moments** (Method of Moments model only)
- **Normalized Soot Aggregation Moments** (Method of Moments model with Aggregation only)
- **Average Number of Particles in Soot Aggregate** (Method of Moments model with Aggregation only)
- **Primary Particle Diameter** (Method of Moments model with Aggregation only)

These parameters are contained in the **Soot...** category of the variable selection drop-down list that appears in postprocessing dialog boxes.

21.4. Using the Decoupled Detailed Chemistry Model

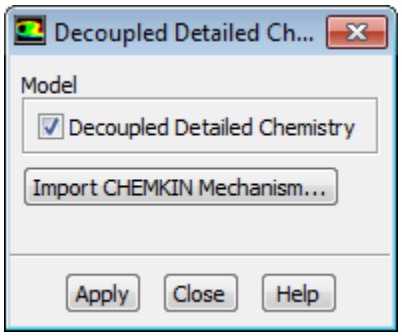
The decoupled detailed chemistry pollutant model is used to postprocess slowly-forming, trace pollutant species on a steady-state flow field using detailed chemical kinetic mechanisms. For theoretical information, refer to Decoupled Detailed Chemistry Model. The recommended procedure is as follows:

1. Calculate your steady combustion problem using the Species Transport, Non-Premixed, Partially-Premixed, or PDF Transport models in ANSYS Fluent. It is recommended that you save your case and data file, as it may be difficult to revert to the original settings after you set up the decoupled detailed chemistry pollutant model.

2. Enable the **Decoupled Detailed Chemistry** option in the **Decoupled Detailed Chemistry** dialog box. To access this dialog box, make sure that either the non-premixed or partially premixed model is selected, or that reactions are enabled if the species transport or PDF transport models are used.

   Setup → Models → Species → Decoupled Detailed Chemistry → Edit...

   Figure 21.19: The Decoupled Detailed Chemistry Dialog Box

3. Click **Import CHEMKIN Mechanism...** to import your detailed chemistry mechanism in CHEMKIN format. The **CHEMKIN Mechanism Import** dialog box (described in Importing a Volumetric Kinetic Mechanism in CHEMKIN Format (p. 967)) will open, where you will browse to the file. After the CHEMKIN mechanism is imported, the **Decoupled Detailed Chemistry** dialog box will expand to include the species.

   The species in the initial detailed chemical kinetic mechanism used to obtain the steady-state solution appear in the **Original Species** list. The species in the detailed chemistry mechanism you have imported appear in the **Pollutant Species** list. Note that if the species is present in both mechanisms, it will be listed only under **Original Species** and will not be available as a pollutant species.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Select the **Pollutant Species** and click **Apply**. The pollutant species are typically slowly forming (far from chemical equilibrium), and occur at miniscule mass fractions. ANSYS Fluent will create a mixture material called **pollutant-mixture** and enable the **Species Transport** model with the **Stiff Chemistry Solver**. All species in the imported mechanism that were not in the original combustion model and are not pollutant species will be calculated by chemical equilibrium at the cell temperature. The solution of all transport equations, except the selected pollutant species, are disabled.

   Note

   Species that are not identified as pollutants and do not participate in the reactions amongst the pollutant species are eliminated. Similarly, reactions in the imported CHEMKIN mechanism, which do not include any pollutant species are also eliminated.

5. Click **Integration Parameters...** to open the **Integration Parameters** dialog box. Set the **ISAT Parameters**, such as the **ISAT Error Tolerance** and **Max. Storage**. An **ISAT Error Tolerance** of 1e-5 is recommended, and **Max. Storage** should be set to a large fraction of the free RAM memory available. To learn more about setting integration parameters, refer to Using ISAT (p. 1128).

6. Define the boundary conditions for all pollutants at flow inlets (these are usually zero).

   Setup → Boundary Conditions

7. Iterate until convergence.

   Solution → Run Calculation

8. Review the mass fractions of pollutants by generating graphical plots or alphanumeric reports in the usual way.

9. Save a new set of case and data files, if desired.

   File → Write → Case & Data...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 22: Predicting Aerodynamically Generated Noise

This chapter provides an overview of ANSYS Fluent's approaches to computing aerodynamically generated sound, the model setup, and the procedure for computing sound. For more information about the underlying theory behind aerodynamically generated sound, see Aerodynamically Generated Noise in the Theory Guide.

22.1. Overview
22.2. Using the Ffowcs-Williams and Hawkings Acoustics Model
22.3. Using the Broadband Noise Source Models

22.1. Overview

Considering the breadth of the discipline and the challenges encountered in aerodynamically generated noise, it is not surprising that a number of computational approaches have been proposed over the years whose sophistication, applicability, and cost widely vary.

ANSYS Fluent offers three approaches to computing aerodynamically generated noise; a direct method, an integral method based on acoustic analogy and a method that utilizes broadband noise source models.

For additional information, see the following sections:

22.1.1. Direct Method
22.1.2. Integral Method Based on Acoustic Analogy
22.1.3. Broadband Noise Source Models

22.1.1. Direct Method

In this method, both generation and propagation of sound waves are directly computed by solving the appropriate fluid dynamics equations. Prediction of sound waves always requires time-accurate solutions to the governing equations. Furthermore, in most practical applications of the direct method, one has to employ governing equations that are capable of modeling viscous and turbulence effects, such as unsteady Navier-Stokes equations (that is, DNS), RANS equations, and filtered equations used in LES and hybrid RANS-LES models.

The direct method is therefore computationally difficult and expensive inasmuch as it requires highly accurate numerics, very fine computational meshes all the way to receivers, and acoustically nonreflecting boundary conditions. The computational cost becomes prohibitive when sound is to be predicted in the far field (for example, hundreds of chord-lengths in the case of an airfoil). The direct method becomes feasible when receivers are in the near field (for example, cabin noise). In many such situations involving near-field sound, sounds (or pseudo-sounds for that matter) are predominantly due to local hydrodynamic pressure, which can be predicted with a reasonable cost and accuracy.

Since sound propagation is directly resolved in this method, you would normally solve the compressible form of the governing equations (for example, compressible RANS equations, compressible form of filtered equations for LES). Only in situations where the flow is low subsonic and the receivers in the near field sense primarily local hydrodynamic pressure fluctuations (that is, pseudo sound) can incom-

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

pressible flow formulations be used. But this incompressible treatment will also not allow you to simulate resonance and feedback phenomena.

22.1.2. Integral Method Based on Acoustic Analogy

For predictions of mid- to far-field noise, the methods based on Lighthill's acoustic analogy [58] (p. 2890) offer viable alternatives to the direct method. In this approach, the near-field flow obtained from appropriate governing equations such as unsteady RANS equations, DES, SAS, SDES, SBES, or LES are used to predict the sound with the aid of analytically derived integral solutions to wave equations. The acoustic analogy essentially decouples the propagation of sound from its generation, allowing one to separate the flow solution process from the acoustics analysis.

ANSYS Fluent offers a method based on the Ffowcs-Williams and Hawkings (FW-H) formulation [30] (p. 2888). The FW-H formulation adopts the most general form of Lighthill's acoustic analogy, and is capable of predicting sound generated by equivalent acoustic sources such as monopoles, dipoles, and quadrupoles. ANSYS Fluent adopts a time-domain integral formulation wherein time histories of sound pressure, or acoustic signals, at prescribed receiver locations are directly computed by evaluating a few surface integrals.

Time-accurate solutions of the flow-field variables, such as pressure, velocity components, and density on source (emission) surfaces, are required to evaluate the surface integrals. Time-accurate solutions can be obtained from unsteady Reynolds-averaged Navier-Stokes (URANS) equations, large eddy simulation (LES), or hybrid RANS-LES models as appropriate for the flow at hand and the features that you want to capture (for example, vortex shedding). The source surfaces can be placed not only on impermeable walls, but also on interior (permeable) surfaces, which enables you to account for the contributions from the quadrupoles enclosed by the source surfaces. Both broadband and tonal noise can be predicted depending on the nature of the flow (noise source) being considered, turbulence model employed, and the time scale of the flow resolved in the flow calculation.

The FW-H acoustics model in ANSYS Fluent allows you to select multiple source surfaces and receivers. It also permits you either to save the source data for a future use, or to carry out an "on the fly" acoustic calculation simultaneously as the transient flow calculation proceeds, or both. Sound pressure signals therefore obtained can be processed using the fast Fourier transform (FFT) and associated postprocessing capabilities to compute and plot such acoustic quantities as the overall sound pressure level (SPL) and power spectra.

One important limitation of ANSYS Fluent's FW-H model is that it is applicable only to predicting the propagation of sound toward *free space*. Therefore, while the model can be legitimately used to predict far-field noise due to external aerodynamic flows, such as the flows around ground vehicles and aircraft, it cannot be used for predicting the noise propagation inside ducts or wall-enclosed space.

22.1.3. Broadband Noise Source Models

In many practical applications involving turbulent flows, noise does not have any distinct tones, and the sound energy is continuously distributed over a broad range of frequencies. In those situations involving *broadband noise*, statistical turbulence quantities readily computable from RANS equations can be utilized, in conjunction with semi-empirical correlations and Lighthill's acoustic analogy, to shed some light on the source of broadband noise.

ANSYS Fluent offers several such *source models* that enable you to quantify the local contribution (per unit surface area or volume) to the total acoustic power generated by the flow. They include the following:

- Proudman's formula

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- jet noise source model
- boundary layer noise source model
- source terms in the linearized Euler equations
- source terms in Lilley's equation

Considering that one would ultimately want to come up with some measures to mitigate the noise generated by the flow in question, the source models can be employed to extract useful diagnostics on the noise source to determine which portion of the flow is primarily responsible for the noise generation. Note, however, that these source models do not predict the sound at receivers.

Unlike the direct method and the FW-H integral method, the broadband noise source models do not require transient solutions to any governing fluid dynamics equations. All the source models need is what typical RANS models would provide, such as the mean velocity field, turbulent kinetic energy (k) and the dissipation rate (ε). Therefore, the use of broadband noise source models requires the least computational resources.

22.2. Using the Ffowcs-Williams and Hawkings Acoustics Model

The procedure for computing sound using the FW-H acoustics model in ANSYS Fluent consists largely of two steps. In the first step, a time-accurate flow solution is generated from which time histories of the relevant variables (for example, pressure, velocity, and density) on the selected source surfaces are obtained. In the second step, sound pressure signals at the user-specified receiver locations are computed using the source data collected during the first step.

Important

Note that you can also use the FW-H model for a steady-state simulation in the case where your model has a single moving reference frame. Here, the thickness and loading noise due to the motion of the noise sources is computed using the FW-H integrals (see Equation 15.5 and Equation 15.6 in the Theory Guide), except that the term involving the time derivative of surface pressure (contribution to $\dot{L}_r$ in Equation 15.6 in the Theory Guide) is set to zero.

In computing sound pressure using the FW-H integral solution, ANSYS Fluent uses a so-called "forward-time projection" to account for the time delay between the emission time (the time at which the sound is emitted from the source) and the reception time (the time at which the sound arrives at the receiver location). The forward-time projection approach enables you to compute sound at the same time "on the fly" as the transient flow solution progresses, without having to save the source data.

In this section, the procedure for setting up and using the FW-H acoustics model is outlined first, followed by detailed descriptions of each of the steps involved. Remember that only the steps that are pertinent to acoustics modeling are discussed here. For information about the inputs related to other models that you are using in conjunction with the FW-H acoustics model, see the appropriate sections for those models.

The general procedure for carrying out an FW-H acoustics calculation in ANSYS Fluent is as follows:

1. Calculate a converged flow solution. For a transient case, run the transient solution until you obtain a "statistically steady-state" solution as described below.
2. Enable the FW-H acoustics model and set the associated model parameters.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → Models → Acoustics Edit...

3. Specify the source surface(s) and choose the options associated with acquisition and saving of the source data. For a steady-state case, specify the rotating surface zone(s) as the source surface(s).

4. Specify the receiver location(s).

5. Continue the transient solution for a sufficiently long period of time and save the source data (transient cases only).

   Solution → Run Calculation

6. Compute and save the sound pressure signals.

   Solution → Run Calculation → Acoustic Signals...

7. Postprocess the sound pressure signals.

   Results → Plots → FFT Edit...

Important

Before you start the acoustics calculation for a transient case, an ANSYS Fluent transient solution should have been run to a point where the transient flow field has become "statistically steady". In practice, this means that the unsteady flow field under consideration, including all the major flow variables, has become fully developed in such a way that its statistics do not change with time. Monitoring the major flow variables at selected points in the domain is helpful for determining if this condition has been met.

As discussed earlier, URANS, LES, and hybrid RANS-LES models are all legitimate candidates for transient flow calculations. For stationary source surfaces, the frequency of the aerodynamically generated sound heard at the receivers is largely determined by the time scale or frequency of the underlying flow. Therefore, one way to determine the time-step size for the transient computation is to make it small enough to resolve the smallest characteristic time scale of the flow at hand that can be reproduced by the mesh and turbulence adopted in your model.

Once you have obtained a statistically stationary flow-field solution, you are ready to acquire the source data.

For additional information, see the following sections:

22.2.1. Enabling the FW-H Acoustics Model
22.2.2. Specifying Source Surfaces
22.2.3. Specifying Acoustic Receivers
22.2.4. Specifying the Time Step
22.2.5. Postprocessing the FW-H Acoustics Model Data
22.2.6. FFT of Acoustic Sources: Band Analysis and Export of Surface Pressure Spectra

22.2.1. Enabling the FW-H Acoustics Model

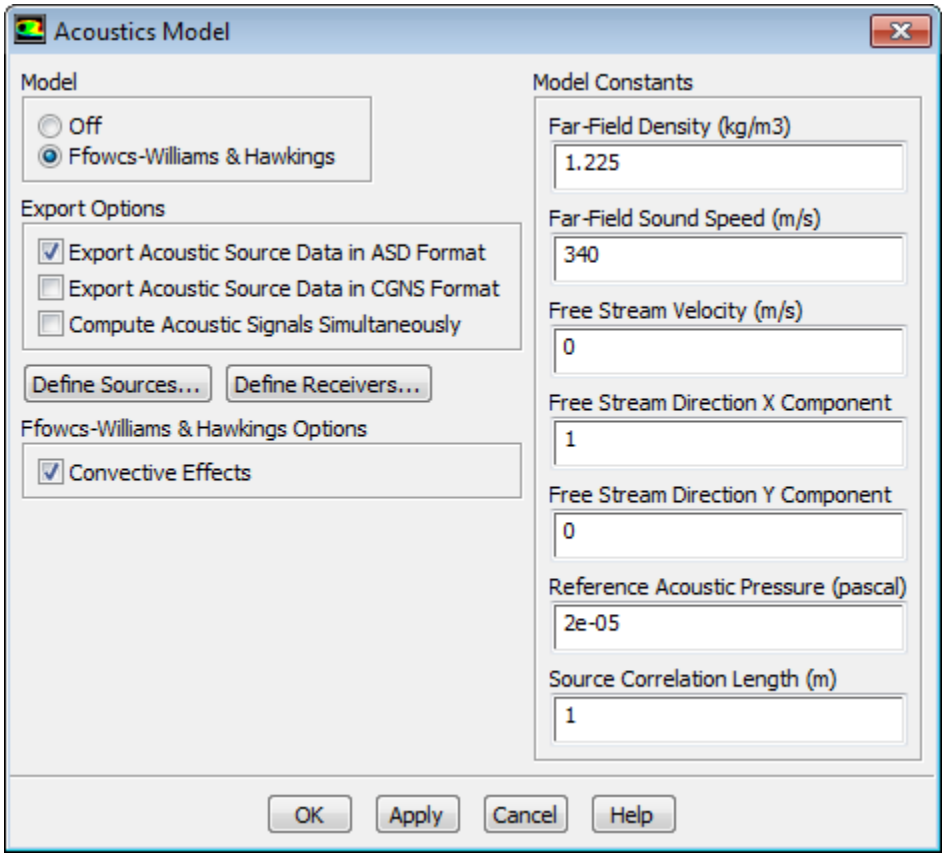
To enable the FW-H acoustics model, select **Ffowcs-Williams & Hawkings** in the **Acoustics Model** dialog box (Figure 22.1: The Acoustics Model Dialog Box (p. 1209)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → Models → Acoustics Edit...

When you select **Ffowcs-Williams & Hawkings**, the dialog box will expand to show the relevant fields for user inputs.

Figure 22.1: The Acoustics Model Dialog Box



22.2.1.1. Setting Model Constants

Under **Model Constants** in the **Acoustics Model** dialog box, specify the relevant acoustic parameters and constants used by the model.

Far-Field Density

(for example, ρ_0 in Equation 15.1 in the Theory Guide) is the far-field fluid density.

Far-Field Sound Speed

(for example, a_0 in Equation 15.1) is the sound speed in the far field (= $\sqrt{\gamma R T_0}$).

Free Stream Velocity and Free Stream Direction
are required when the convective effects are taken into account. They become available in the interface when **Convective Effects** is enabled.

> **Important**
>
> The use of **Convective Effects** with the proper **Free Stream Velocity** and **Free Stream Direction** is strongly recommended for all cases dealing with external flows around bodies.

Reference Acoustic Pressure
(for example, p_{ref} in Equation 31.41 (p. 1880)) is used to calculate the sound pressure level in dB (see Using the FFT Utility (p. 1875)). The default reference acoustic pressure is 2×10^{-5} Pa.

Number of Time Steps Per Revolution
is available only for steady-state cases that have a single moving reference frame. Here you will specify the number of equivalent time steps that it will take for the rotating zone to complete one revolution.

Number of Revolutions
is available only for steady-state cases that have a single moving reference frame. Here you will specify the number of revolutions that will be simulated in the model.

Source Correlation Length
is required when sound is to be computed using a 2D flow result. The FW-H integrals will be evaluated over this length in the depth-wise direction using the identical source data (see Figure 22.2: The Acoustics Model Dialog Box for a 2D Steady-State Case with a Single Moving Reference Frame (p. 1212)).

The default values are appropriate for sound propagating in air at atmospheric pressure and temperature.

22.2.1.2. Computing Sound "on the Fly"

The FW-H acoustics model in ANSYS Fluent allows you to perform simultaneous calculation of the sound pressure signals at the prescribed receivers without having to write the source data to files, which can save a significant amount of disk space on your machine. To enable this "on-the-fly" calculation of sound, enable the **Compute Acoustic Signals Simultaneously** option in the **Acoustics Model** dialog box.

> **Important**
>
> Because the noise computation takes a negligible percentage of memory and computational time compared to a transient flow calculation, this option can be used by itself or along with the process of source data file export and sound calculation. For the latter, computing signals "on the fly" allows you to see when the signals have become statistically steady so you can know when to stop the simulation.

When the **Compute Acoustic Signals Simultaneously** option is enabled, the ANSYS Fluent console window will print a message at the end of each time step indicating that the sound pressure signals have been computed (for example, `Computing sound signals at x receiver locations...`, where `x` is the number of receivers you specified). Enabling this option instructs ANSYS

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fluent to compute sound pressure signals at the end of each time step, which will slightly increase the computation time.

Important

Note that this option is available only when the FW-H acoustics model has been enabled. See below for details about exporting source data without enabling the FW-H model.

22.2.1.3. Writing Source Data Files

Although the "on-the-fly" capability is a convenient feature, you will want to save the source data as well, because the acquisition of source data during a transient flow-field calculation is the most time-consuming part of acoustics computations, and you most likely will not want to discard it. By saving the source data, you can always reuse it to compute the sound pressure signals at new or additional receiver locations.

To save the source data to files, enable either the **Export Acoustic Source Data in ASD Format** or the **Export Acoustic Source Data in CGNS Format** option, or both. After you have made your selection, the relevant source data at all face elements of the selected source surfaces will be written into the files you specify. The source data vary depending on the solver option you have chosen and whether the source surface is a wall or not. Table 22.1: Source Data Saved in Source Data Files (p. 1212) shows the flow variables saved as the source data.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 22.2: The Acoustics Model Dialog Box for a 2D Steady-State Case with a Single Moving Reference Frame

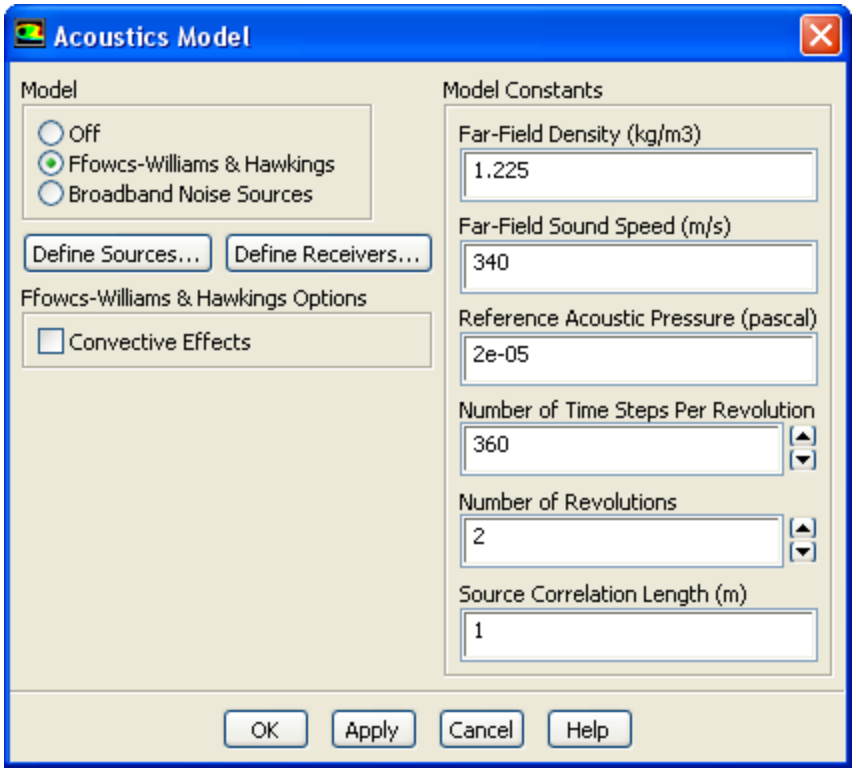


Table 22.1: Source Data Saved in Source Data Files

Solver Option	Source Surface	Source Data
incompressible	walls	p
incompressible	permeable surfaces	p,u,v,w
compressible	walls	p
compressible	permeable surfaces	ρ,p,u,v,w

See Specifying Source Surfaces (p. 1214) for details on how to specify parameters for exporting source data.

22.2.1.3.1. Exporting Source Data Without Enabling the FW-H Model: Using the ANSYS Fluent ASD Format

You can export sound source data for use with SYSNOISE without having to enable the Ffowcs-Williams and Hawkings (FW-H) model. You still must specify source surfaces (see Specifying Source Surfaces (p. 1214)), as `.index` and `.asd` files are required by SYSNOISE. In addition, you can choose fluid zones as emission sources if you want to export quadrupole sources. To enable the selection of fluid zones as sources, use the

`define` → `models` → `acoustics` → `export-volumetric-sources?`

text command and change the selection to `yes`.

SYSNOISE also requires centroid data for source zones that are being exported.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For fan noise calculations, once you have specified the source zones in the **Acoustic Sources** dialog box and you have selected **Export Acoustic Source Data in ASD Format** from the **Acoustics Model** dialog box, you can export geometry in cylindrical coordinates by using the

`define` → `models` → `acoustics` → `cylindrical-export?`

text command and changing the selection to `yes`. By default, ANSYS Fluent exports source zones for SYSNOISE in Cartesian coordinates.

You can then export the centroid data to a data file using the following text command:

`define` → `models` → `acoustics` → `write-centroid-info`

Since you will not be using the FW-H model to compute signals, you will not need to specify any acoustic model parameters or receiver locations. Also, you will not be able to enable the **Compute Acoustic Signals Simultaneously** option in the **Acoustics Model** dialog box, and **Acoustic Signals...** will not be available in the **Run Calculation** task page and in the menu for the **Run Calculation** tree item.

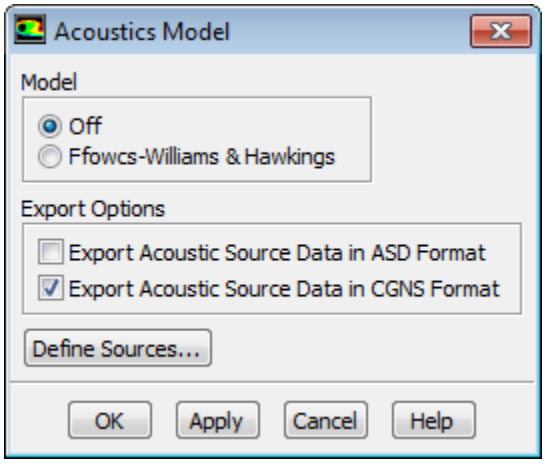
22.2.1.3.2. Exporting Source Data Without Enabling the FW-H Model: Using the CGNS Format

The sound source data for non-permeable surfaces can be exported in the CGNS file format (for Virtual Lab) without having to enable the Ffowcs-Williams and Hawkings (FW-H) model. Enable the **Export Acoustic Source Data in CGNS Format** option in the **Acoustics Model** dialog box (Figure 22.3: The Acoustics Model Dialog Box for Exporting in CGNS Format (p. 1213)). Specify the source surfaces in the **Acoustics Sources** dialog box (see Specifying Source Surfaces (p. 1214)) where, by default, the **Number of Time Steps per File** is set to `1`.

Note

ANSYS Fluent does not support exporting acoustic source data in CGNS format for polyhedral meshes.

Figure 22.3: The Acoustics Model Dialog Box for Exporting in CGNS Format



Virtual Lab requires a mesh data file (named `<prefix>_mesh.cgns`) and a solution data file (named `<prefix>_<timestep>.cgns`). The string `<prefix>` is a generic name, which you will specify in the **Source Data Root File Name** in the **Acoustics Sources** dialog box. There is one single solution

data file (`.cgns`) per time level exported, which contains the static pressure at the wall-face centroid location. The `.cgns` files will be stored in a directory, which you specify (named `<directory_name>/<prefix>`) in the **Source Data Root File Name**.

In addition, you can export quadrupole sources data by choosing fluid zones as emission sources. To enable the selection of fluid zones as sources, use the text command:

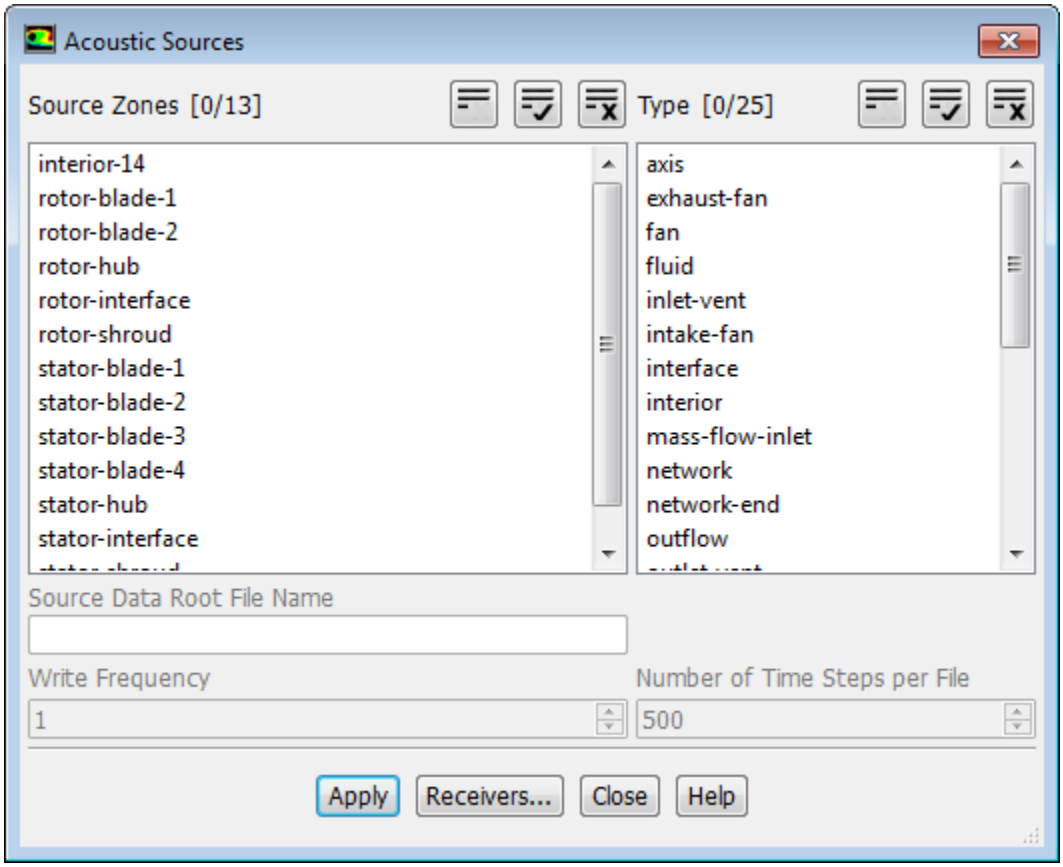
`define` → `models` → `acoustics` → `export-volumetric-sources-cgns?`

When asked if you would like to `Export volumetric sources?` enter `yes`. Note that Virtual Lab requires volumetric mesh data file (`<prefix>_Q_mesh.cgns`) and quadrupole solution data files (`<prefix>_Q_<timestep>.cgns`). The `.cgns` file will be stored in a similar way to that of dipole data export, in the directory specified by you in the **Source Data Root File Name** text entry box.

22.2.2. Specifying Source Surfaces

In the **Acoustics Model** dialog box, click the **Define Sources...** button to open the **Acoustic Sources** dialog box (Figure 22.4: The Acoustics Model Dialog Box (p. 1214)). Here you will specify the source surface(s) to be used in the acoustics calculation and the inputs associated with saving source data to files.

Figure 22.4: The Acoustics Model Dialog Box



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Under **Source Zones**, you can select multiple emission (source) surfaces and the surface **Type** that you can select is *not* limited to a **wall**. You can also choose **interior** surfaces and sliding interfaces (both stationary and rotating) as source surfaces.

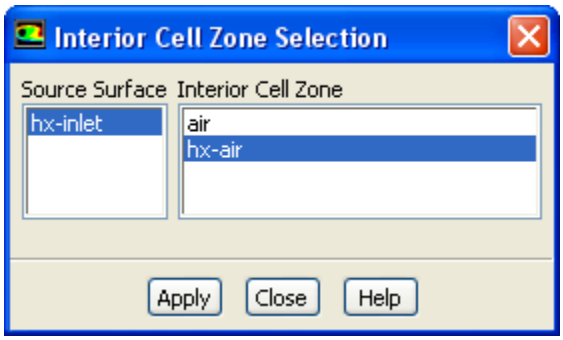
Important

The ability to choose multiple source surfaces is useful for investigating the contributions from individual source surfaces. The results based on the use of multiple source surfaces are valid as long as there are negligible acoustic interactions among the surfaces. Therefore, some caution should be taken when selecting multiple source surfaces.

In cases where multiple source surfaces are selected, no source surface may enclose any of the other source surfaces. Otherwise, the sound pressure calculated based on the source surfaces will not be accurate, as the contribution from the enclosed (inner) source surfaces is over predicted, since the FW-H model is unable to account for the shading of the sound from the inner source surfaces by the enclosure surface.

If you specify any interior surfaces as source surfaces, the interior surface must be generated in advance (for example, in GAMBIT) in such a way that the two cell zones adjacent to the surface have different cell zone IDs. Furthermore, you must correctly specify which of the two zones is occupied by the quadrupole sources (interior cell zone). This will allow ANSYS Fluent to determine the direction in which the sound will propagate. When you first attempt to select a legitimate interior surface (that is, an interior surface having two different cell zones on both sides) as a source surface, the Interior Cell Zone Selection Dialog Box (p. 2294) (Figure 22.5: The Interior Cell Zone Selection Dialog Box (p. 1215)) will appear. You must then select the interior cell zones from the two zones listed under the **Interior Cell Zone**. Figure 22.6: An Interior Source Surface (p. 1216) shows an example of an interior source surface.

Figure 22.5: The Interior Cell Zone Selection Dialog Box



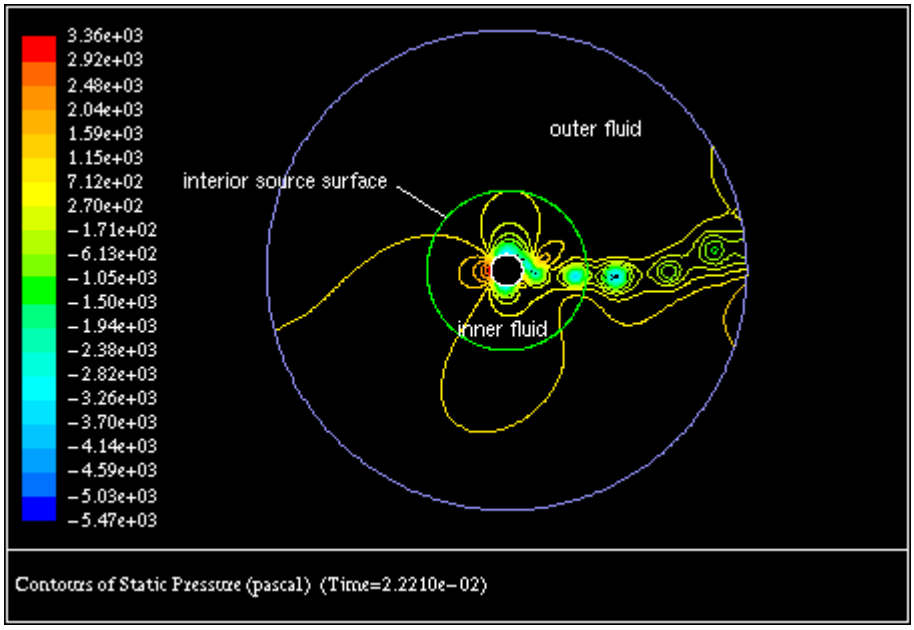
Like general interior surfaces, if the source surfaces selected are sliding interfaces, a dialog box similar to Figure 22.5: The Interior Cell Zone Selection Dialog Box (p. 1215) will appear that will show the two adjacent cell zones and you will be asked to specify the zone that has the sound sources.

Important

When a permeable surface (either interior or sliding interface) is chosen as the source surface, other wall surfaces inside the volume enclosed by the permeable surface that generate sound should not be chosen for the acoustics calculation. For example, when running an "on-the-fly" calculation, if both these surfaces are selected, the sound pressure will be counted twice.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 22.6: An Interior Source Surface



22.2.2.1. Saving Source Data

To save the source data, you have to specify the **Source Data Root File Name**, **Write Frequency** (in number of time steps), and **Number of Time Steps per File** in the **Acoustic Sources** dialog box.

The **Source Data Root File Name** is used to give the names of the source data files (for example, `acoustic_examplexxxx.asd`, where `xxxx` is the global time-step index of the transient solution) and an index file (for example, `acoustic_example.index`) that will store the information associated with the source data. The **Write Frequency** allows you to control how often the source data will be written. This will enable you to save disk space if the time-step size used in the transient flow simulation is smaller than necessary to resolve the sound frequency you are attempting to predict. In most situations, however, you will want to save the source data at every time step and use the default value of `1`.

Since acoustics calculations usually generate thousands of time steps of source data, you may want to split the data into several files. Specifying the **Number of Time Steps per File** allows you to write the source data into separate files for different simulation intervals, the duration of which (in terms of the number of transient flow time steps) is specified by you. For example, if you specify 100 for this parameter, each file will contain source data for an interval length of 100 time steps regardless of the write frequency.

You will find this feature useful if you want to use a selected number of source data files to compute the sound pressure rather than using all the data. For example, you may want to exclude an initial portion of the source data from your acoustics calculation because you may realize later that the flow field has not fully attained a statistically steady state.

After you click **Apply**, ANSYS Fluent will create the index file (for example, `acoustics_example.index`), which contains information about the source data.

Important

If you choose to save source data, keep in mind that the source data can use up a considerable amount of disk space, especially if the mesh being used has a large number of face elements

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

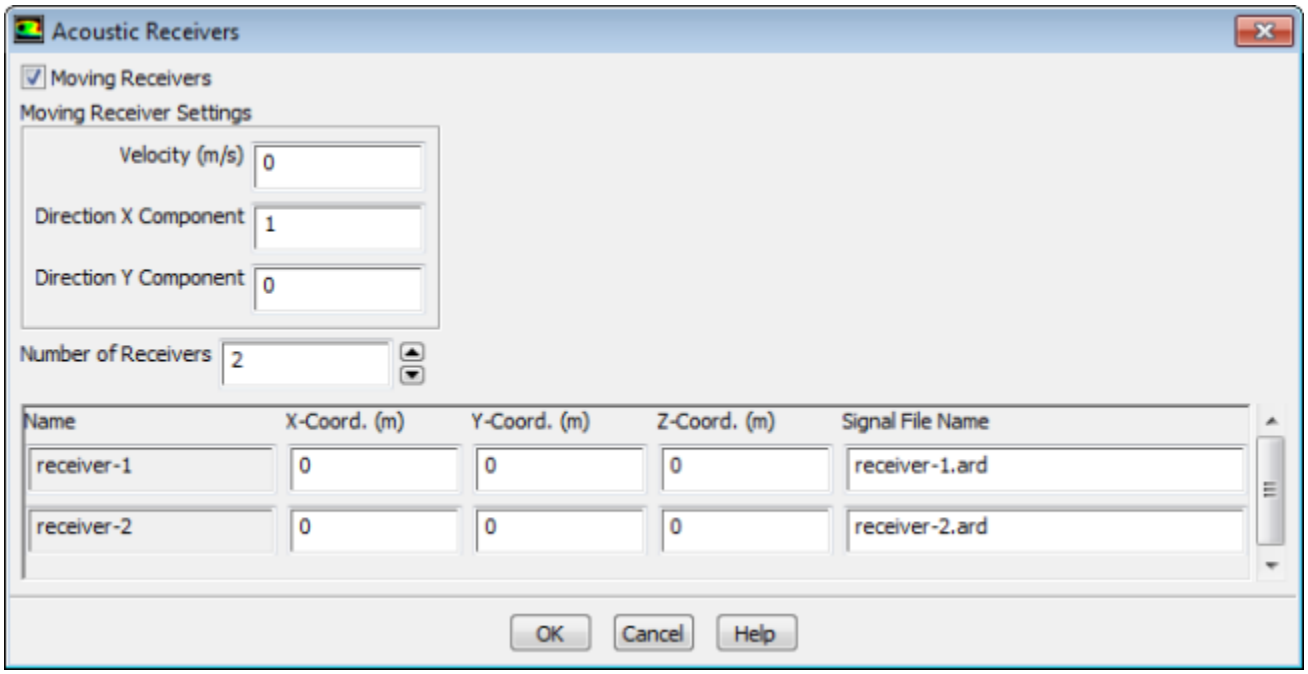
on the source surfaces you selected. ANSYS Fluent will print out the disk space requirement per time step at the time of source surface selection if the **Export Acoustic Source Data in ASD Format** option is enabled in the **Acoustics Model** dialog box.

At this point, if you have chosen to perform your acoustics calculation in two steps, (that is, saving the source data first, and computing the sound at a later time), you can go ahead and instruct ANSYS Fluent to perform a suitable number of time steps, and the source data will be saved to the disk. If you chose to perform an "on-the-fly" acoustic calculation, then you must specify receiver locations (see Specifying Acoustic Receivers (p. 1217)) before you run the unsteady ANSYS Fluent solution any further.

22.2.3. Specifying Acoustic Receivers

In the **Acoustics Model** dialog box, click the **Define Receivers...** button to open the **Acoustic Receivers** dialog box (Figure 22.7: The Acoustic Receivers Dialog Box (p. 1217)).

Figure 22.7: The Acoustic Receivers Dialog Box



Important

Note that you can also open the **Acoustic Receivers** dialog box by clicking the **Receivers...** button in the **Acoustic Sources** or the **Acoustic Signals** dialog box.

If required, you can enable **Moving Receivers** to specify the receiver motion. In this case, the defined **Velocity** magnitude and **Direction** apply to all receivers. The receiver locations, defined below, are then interpreted as the starting locations at the time that sound emission originates. The origination of sound emission is determined as follows:

- For "on the Fly" use of the FW-H model (see Computing Sound "on the Fly" (p. 1210)) the sound emission time is counted from the physical time of activation of the **Compute Acoustic Signals Simultaneously** option (see Figure 22.1: The Acoustics Model Dialog Box (p. 1209)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When the FW-H model reads the previously written acoustic source data (see Reading Unsteady Acoustic Source Data (p. 1220)), the sound emission time starts from the physical time associated with the first selected source data file.

Important

When using moving receivers, note that if you start from a different source data file, this results in different receiver locations due to the different emission time origin associated with the different source data file.

Increase the **Number of Receivers** to the total number of receivers for which you want to compute sound, and enter the coordinates for each receiver in the **X-Coord.**, **Y-Coord.**, and **Z-Coord.** fields. Note that because ANSYS Fluent's acoustics model is ideally suited for far-field noise prediction, the receiver locations you define should be at a reasonable distance from the sources of sound (that is, the selected source surfaces). The receiver locations can also fall outside of the computational domain.

For each receiver, you can specify a file name in the **Signal File Name** field. These files will be used to store the sound pressure signals at the corresponding receivers. By default, the files will be named `receiver-1.ard`, `receiver-2.ard`, and so on.

Once the receiver locations have been defined, the setup for your acoustic calculation is complete.

22.2.4. Specifying the Time Step

When using an implicit-in-time solution algorithm (dual-time stepping), and depending on the physical time step size and the most important time scales in the flow, you can write the acoustic source data at every time step. You can also coarsen it in time by a given frequency factor. The highest possible frequency the acoustic analysis can generate is based on the time step size of the collected acoustic source data.

When using the **Density-Based** explicit solver, with the **Explicit Transient Formulation** selected in the **Solution Methods** task page, the physical time step must be computed by the solver, based on the CFL condition (Courant number). Due to the possibly large fluctuations of the physical time step, an adapting time-stepping procedure can be used when the FW-H acoustics model is enabled. This allows you to use a user-specified time interval for sampling the acoustic data. In turn, the solver adapts its time step, when necessary, without violating the CFL conditions to make sure that data is available at the desired time interval (hence, avoiding data interpolations).

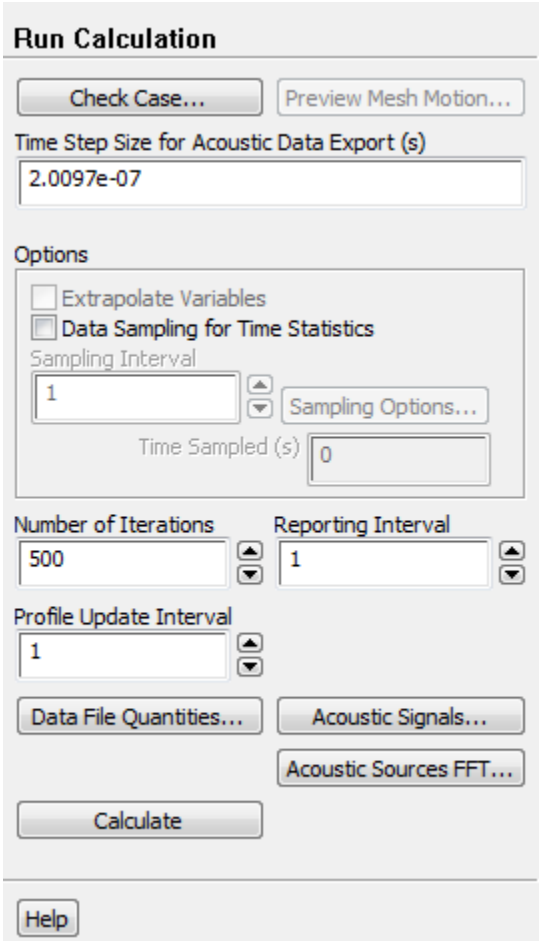
In the **Run Calculation** task page (Figure 22.8: The Run Calculation Task Page (p. 1219)), enter the **Time Step Size for Acoustic Data Export** to specify the time interval for acoustic data sampling. The value of this constant time step size determines the highest frequency that the acoustic analysis reproduces.

You can refer to Performing Time-Dependent Calculations (p. 1608) for more information about the **Run Calculation** task page.

Solution → **Run Calculation**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 22.8: The Run Calculation Task Page



You can now proceed to instruct ANSYS Fluent to perform a transient calculation for a suitable number of time steps. When the calculation is finished, you will have either the source data saved on files (if you chose to save it to a file or files), or the sound pressure signals (if you chose to perform an acoustic calculation "on the fly"), or both (if you chose to save the source data to files *and* if you chose to perform the acoustic calculation "on the fly").

If you chose to save the source data to files, the ANSYS Fluent console window will print a message at the end of each time step indicating that source data have been written (or appended to) a file (for example, `acoustic_example240.asd`).

22.2.5. Postprocessing the FW-H Acoustics Model Data

At this point, you will have either the source data saved to files or the sound pressure signals computed, or both. You can process these data to compute and plot various acoustic quantities using ANSYS Fluent's FFT capabilities. See Fast Fourier Transform (FFT) Postprocessing (p. 1873) for more information.

22.2.5.1. Writing Acoustic Signals

If you chose to perform the acoustic calculation "on the fly", you must write the sound pressure data to files. To do so, select **Write Acoustic Signals** under **Options** in the **Acoustic Signals** dialog box (Figure 22.9: The Acoustic Signals Dialog Box (p. 1220)) and then click **Write**. The computed acoustic pressure will be saved from internal buffer memory into a separate file for each receiver you defined in the **Acoustic Receivers** dialog box (for example, `receiver-1.ard`).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

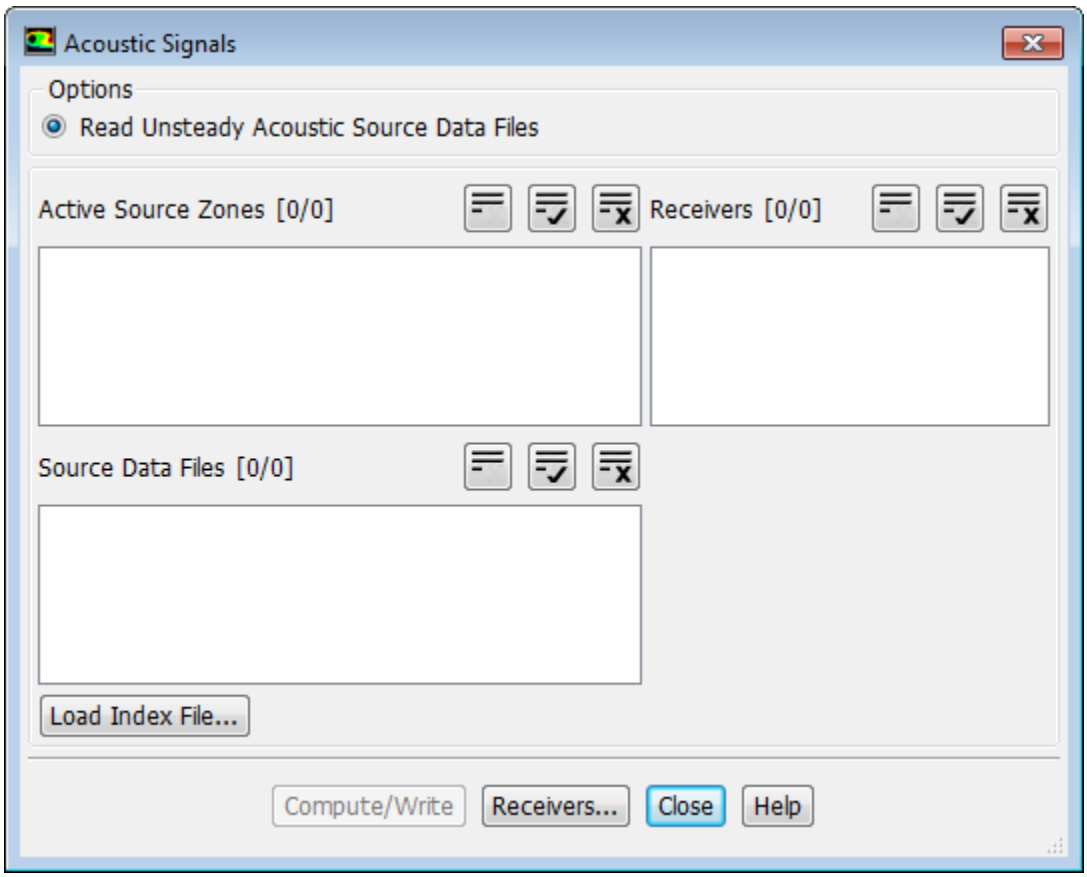
Solution → **Run Calculation** → **Acoustic Signals...**

22.2.5.2. Reading Unsteady Acoustic Source Data

Computing the sound pressure signals using the source data saved to files is done in the **Acoustic Signals** dialog box (Figure 22.9: The Acoustic Signals Dialog Box (p. 1220))

Solution → **Run Calculation** → **Acoustic Signals...**

Figure 22.9: The Acoustic Signals Dialog Box



To compute the sound data, use the following procedure:

1. In the **Acoustic Signals** dialog box, select **Read Unsteady Acoustic Source Data Files** under **Options**.
2. Click **Load Index File...** and select the index file for your computation in the **Select File** dialog box. The file will have the name you entered in the **Source Data Root File Name** field in the **Acoustic Sources** dialog box, followed by the `.index` suffix (for example, `acoustic_example.index`).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. In the **Source Data Files** list, select the source data files that you want to use to compute sound. Source data files will all contain the specified root file name followed by the suffix `.asd`.

   Important

   You can use any number of source data files. However, note that you should select only consecutive files.

4. In the **Active Source Zones** list, select the source zones you want to include to compute sound. See Specifying Source Surfaces (p. 1214) for details about proper source surface selection.

5. In the **Receivers** list, select the receivers for which you want to compute and save sound.

   Optionally, you can click the **Receivers...** button to open the **Acoustic Receivers** dialog box and define additional receivers.

6. Click the **Compute/Write** button to compute and save the sound pressure data. One file will be saved for each receiver you previously specified in the **Acoustic Receivers** dialog box (for example, `receiver-1.ard`).

   Important

   If you enabled both the **Export Acoustic Source Data in ASD Format** and **Compute Acoustic Signals Simultaneously** options in the **Acoustics Model** dialog box, you must first select the **Write Acoustic Signals** option in the **Acoustic Signals** dialog box after the flow simulation has been completed. If you select the **Read Unsteady Acoustic Source Data Files** before writing out the "on-the-fly" data in such a case, the data will be flushed out of the internal buffer memory. To avoid such a loss of data, you should save the ANSYS Fluent case and data files whenever you begin to do an acoustic computation in the **Acoustic Signals** dialog box. The sound pressure data calculated "on the fly" will then be saved into the `.dat` file. Finally, after the "on-the-fly" data is saved, make sure to change the file names of the receivers before doing a sound pressure calculation with the **Read Unsteady Acoustic Source Data Files** option enabled, to avoid overwriting the "on-the-fly" signal files.

   Important

   Note that you can compute and write sound pressure signals only when the FW-H acoustics model has been enabled. See Exporting Source Data Without Enabling the FW-H Model: Using the ANSYS Fluent ASD Format (p. 1212) for details about exporting source data (for example, for SYSNOISE) without enabling the FW-H model.

22.2.5.2.1. Pruning the Signal Data Automatically

Before the computed sound pressure data at each receiver is saved, it is by default automatically pruned. Pruning of the receiver data means clipping the tails of the signal where incomplete source information is available.

The acoustic source data is tabulated from time τ_0 to τ_n. Without auto-pruning, the receiver register begins receiving the earliest sound pressure signal at

$$t_0 = \tau_0 + \frac{r_{min}}{a_0} \tag{22.1}$$

where r_{min} is the shortest distance between the source surfaces and the receiver. However, the receiver will not receive the sound pressure signal from the farthest point on the source surfaces (r_{max}) until the receiver time becomes

$$t_1 = \tau_0 + \frac{r_{max}}{a_0} \tag{22.2}$$

From time t_0 to t_1, the sound accumulated on the receiver register does not include the contribution from the entire source surface area, and therefore the sound pressure data received during that time is not complete. The same thing occurs during the period from

$$t_m = \tau_m + \frac{r_{min}}{a_0} \tag{22.3}$$

to

$$t_n = \tau_n + \frac{r_{max}}{a_0} \tag{22.4}$$

Therefore, pruning means clipping the signal on the incomplete ends, from t_0 to t_1 and t_m to t_n. Auto-pruning can be disabled using the `define` → `models` → `acoustics` → `auto-prune` text command. Although auto-pruning can be disabled, it is expected that you will use only the complete sound pressure data.

22.2.5.3. Reporting the Static Pressure Time Derivative

The RMS value of the static pressure time derivative ($\partial p / \partial t$) is available for postprocessing only on wall surfaces, which are at the same time sources of sound, when the FW-H acoustics model is used.

You can select **Surface dpdt RMS** in the **Acoustics...** category only when you specify at least one wall surface, which is also marked as an acoustic source, in the relevant postprocessing dialog boxes.

22.2.5.4. Using the FFT Capabilities for Sound Pressure Signals

Once the sound pressure signals are computed and saved in files, the sound data is ready to be analyzed using ANSYS Fluent's FFT tools. In the **Fourier Transform** dialog box (Figure 31.89: The Fourier Transform Dialog Box (p. 1876)), click **Load Input File...** and select the appropriate `.ard` file. If the receiver data is still in ANSYS Fluent's memory, then it can directly be processed using the **Process Receiver** option. See Fast Fourier Transform (FFT) Postprocessing (p. 1873) for more information on ANSYS Fluent's FFT capabilities.

Results → **Plots** → **FFT** **Edit...**

22.2.6. FFT of Acoustic Sources: Band Analysis and Export of Surface Pressure Spectra

During a transient flow simulation, the time histories of the instant surface pressure values can be written into the binary acoustic source data (ASD) files. This operation is performed for all mesh cell faces of those surface zones that are selected as acoustic sources. The primary purpose of the ASD files is to supply input data for the Ffowcs-Williams and Hawkings acoustic solver (see Reading Unsteady Acoustic Source Data (p. 1220)). Additionally, these files can be used to visualize the spectral properties of the stored flow pressure signals. The supported spectral properties are the complex amplitudes for

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the individual Fourier modes, and the surface pressure level (SPL) values for the frequency bands, in decibels. Both the constant width and the proportional width bands (octaves and 1/3 octaves) can be specified.

Note

Despite the use of the common acronym "SPL", the calculated values characterize the transient flow pressure on a surface, and not the acoustic pressure; that is, in this context, the acronym refers to the "Surface Pressure Level" (as mentioned previously) and not the "Sound Pressure Level".

In addition to the visualization capabilities, with ANSYS Fluent you can also use the ASD files to export the computed spectrum fields of the flow pressure as CGNS files. The primary purpose of such files is to allow you to use the surface spectrum fields as distributed excitation forces when performing a harmonic response analysis with ANSYS Mechanical. The ability to import CGNS files is available in ANSYS Mechanical to match this export feature in Fluent; for details, see **FLUREAD** in the separate Mechanical APDL *Command Reference*. Using the acoustic wave equation solver of ANSYS Mechanical, it is possible to perform frequency domain simulations for applications such as the vehicle cabin noise caused by external turbulent flow.

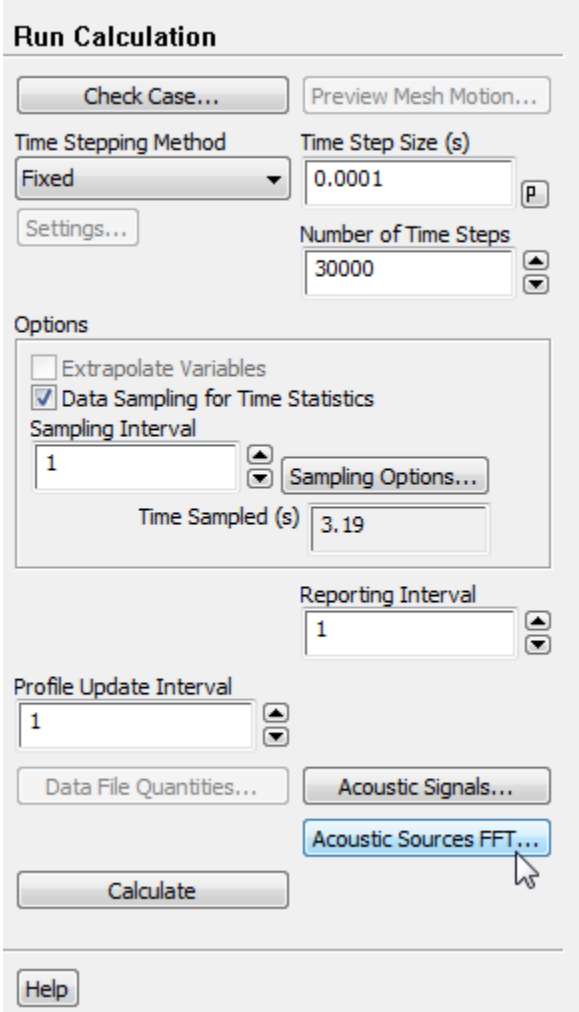
22.2.6.1. Using the FFT of Acoustic Sources

The **Acoustic Sources FFT…** button is available in the **Run Calculation** task page (as shown in Figure 22.10: The Run Calculation Task Page with the Acoustic Sources FFT… Button (p. 1224)) under the following conditions:

- The simulation is set up as a three-dimensional transient case.
- The **Export Acoustic Source Data in ASD Format** option is enabled in the **Acoustics Model** dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 22.10: The Run Calculation Task Page with the Acoustic Sources FFT... Button



Clicking the **Acoustic Sources FFT...** button opens the **Acoustic Sources FFT** dialog box, which has four tabs (each of which is shown in the figures that follow):

- **Read ASD Files**
- **Compute FFT Fields**
- **FFT Surface Variables**
- **Write CGNS Files**

The first two tabs are obligatory, because the actions performed there prepare the Fourier spectra for the actions that follow, including creation of the surface variables to visualize (performed in the third tab) and/or CGNS export (performed in the fourth tab).

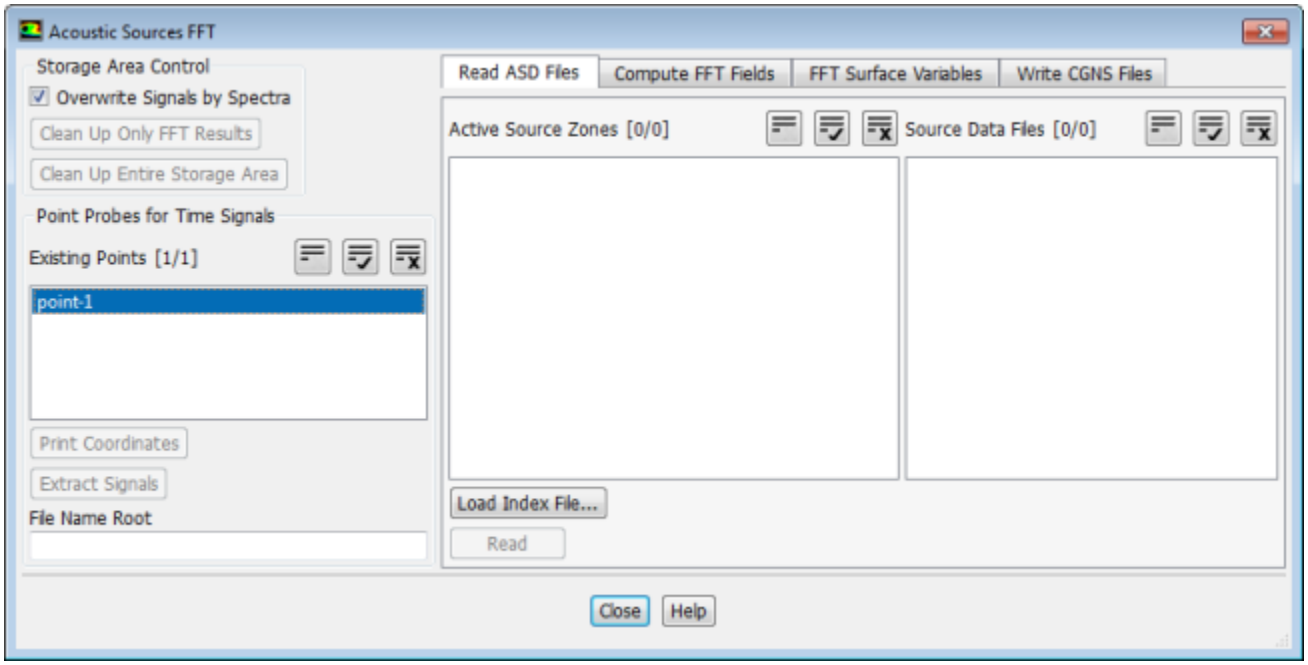
Postprocessing begins in the **Read ASD Files** tab. The **Active Source Zones** selection list shows all of the face zones for which transient export to ASD files has been performed (see Figure 22.11: The Read ASD Files Tab of the Acoustic Source FFT Dialog Box (p. 1225)). The **Source Data Files** selection list shows all of the available ASD files. Note that you can update this information by clicking the **Load Index File...** button and specifying the name of the index file to be read; the index file was created during the ASD files export, and is an ASCII file containing a list of the source zones and a list of the ASD file names. Select in the selection lists the desired face zones as well as the desired ASD files from which

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

you want to read, and click the **Read** button. The console displays information about the data, such as the following example:

```
Overall 10000 timesteps have been read
from 3.0001 s to 4 s
Time step is 0.0001 s
```

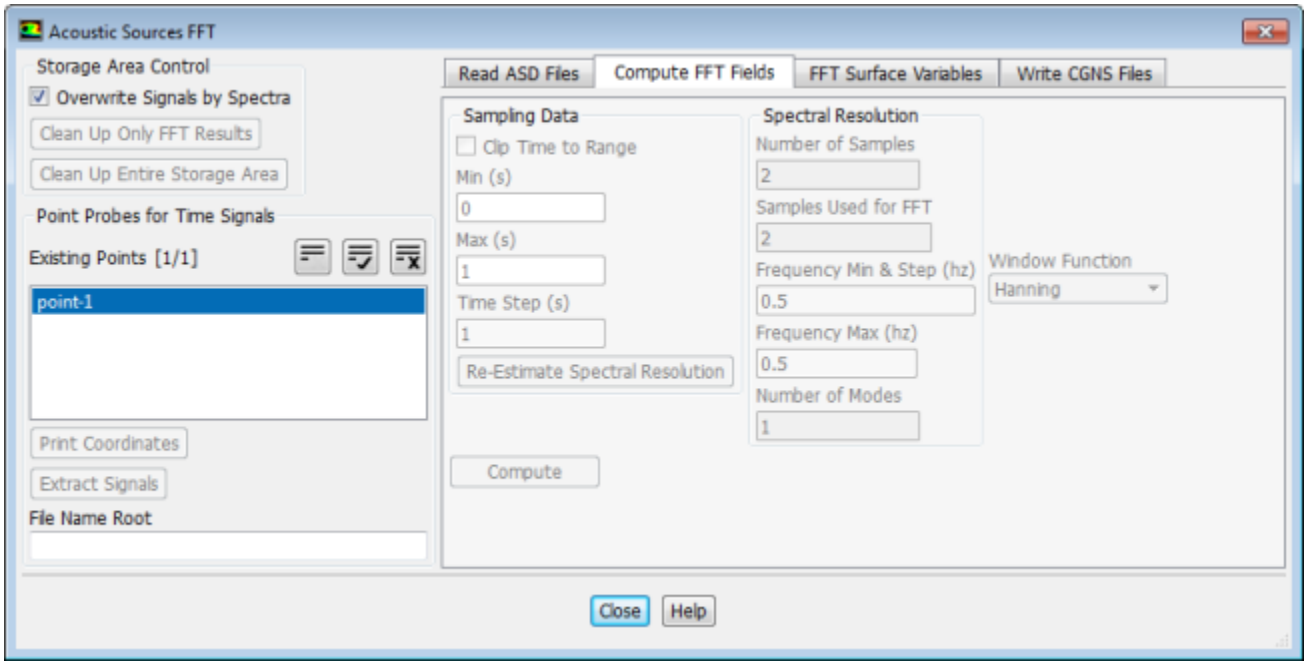
Figure 22.11: The Read ASD Files Tab of the Acoustic Source FFT Dialog Box



Having read the pressure histories, open the **Compute FFT Fields** tab (Figure 22.12: The Compute FFT Fields Tab of the Acoustic Source FFT Dialog Box (p. 1226)). The values shown in the **Spectral Resolution** group box provide information about the statistical properties, which depend on the sampling rate (that is, the simulation time step) and the signal length (that is, the simulation time). These values will change if you enable the **Clip Time to Range** option in the **Sampling Data** group box, reduce the time range for the signals by changing the **Min** and/or **Max** fields, and click the **Re-Estimate Spectral Resolution** button. The **Window Function** selection list allows you to specify a window function using the same choices that are available for plotting the single signal FFT in the **Plot/Modify Input Signal** dialog box. The default settings use the full signals and the Hanning window function. The FFT calculation begins when you click the **Compute** button.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 22.12: The Compute FFT Fields Tab of the Acoustic Source FFT Dialog Box



The fields of the pressure histories and the computed fields of the Fourier spectra are stored in a large memory array, referred to here as the "storage area". This memory is allocated by Fluent dynamically when you read the pressure signals from the ASD files. If you read data for all face zones available in the ASD files, and if they are the wall zones, then the size of the allocated storage area is equal to the total size of the ASD files being read. Since the same or a slightly smaller amount of data is produced by the FFT algorithm, the same storage area can be re-used to store the Fourier spectra without allocating additional memory. Alternatively, the pressure histories can be left intact in memory, and for their spectra the storage area is correspondingly increased.

These two modes of memory allocation are controlled by the **Overwrite Signals by Spectra** option in the **Storage Area Control** group box. When the **Overwrite Signals by Spectra** option is enabled, the Fourier spectra displace in memory the original time signals, which are not kept after the FFT has been computed. Therefore, if you want to re-compute the FFT using the **Clip Time to Range** tool or a different **Window Function** (which are both in the **Compute FFT Fields** tab), you have to first de-allocate the storage area by clicking the **Clean Up Entire Storage Area** button, and then re-read the pressure histories from the ASD files. When the **Overwrite Signals by Spectra** option is disabled, you can re-compute the FFT without re-reading the pressure histories, as many times as you want. To do this, click the **Clean Up Only FFT Results** button, and proceed with computing spectra from the available pressure histories. De-allocation of the storage area (which can be very large) is recommended when you are done using the **Acoustic Sources FFT** dialog box and plan to either proceed to other postprocessing work or continue your transient simulation without re-starting Fluent.

When the pressure histories are read and kept in the storage area, you can create point probes within the surface zones that have been read and extract the single pressure time signals at these probes. The extracted signals and their spectra may help to analyze the transient flow properties. For example, by creating an XY plot of a signal, you can figure out the need for clipping of the sampling time; by creating an XY plot of its spectrum, you can detect the presence of the tonal noise components.

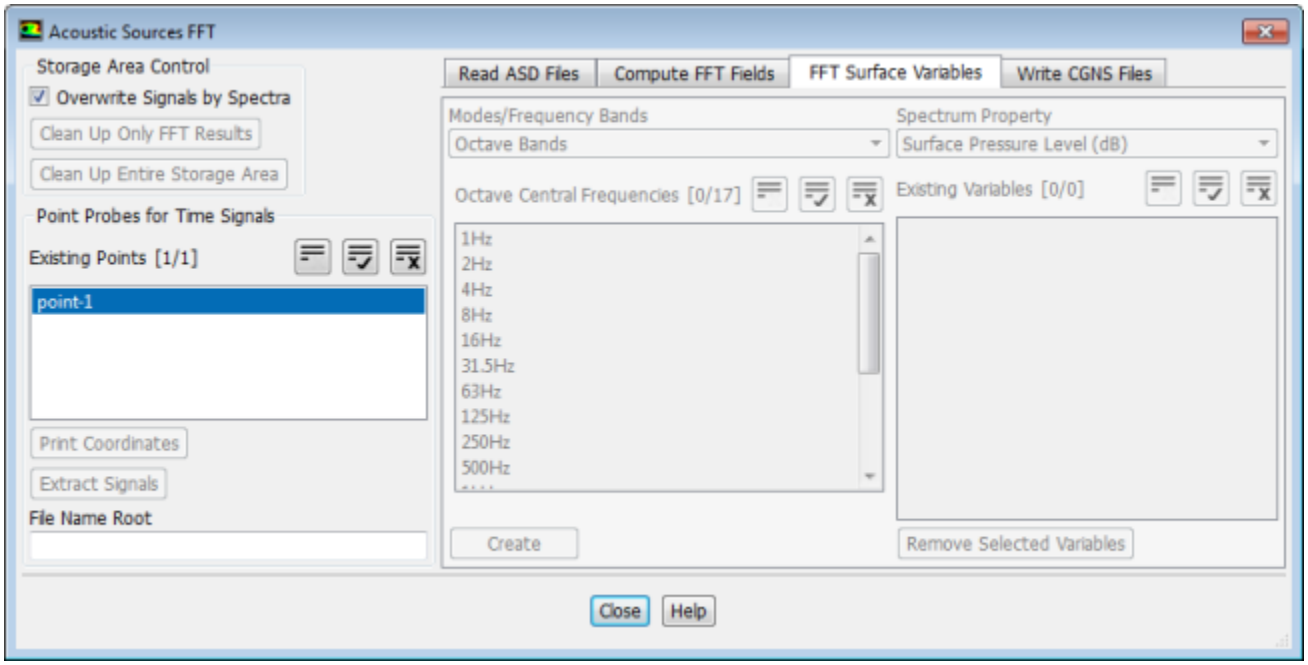
To create a point probe, open the Point Surface Dialog Box (p. 2780) by clicking **Create** and selecting **Point...** in the **Setting Up Domain** ribbon tab (**Surface** group box), and create a point surface by any available method (for example, by specifying the coordinates). The point probe will be created at the face center that is nearest to the point surface on the read surface zones. The created point probes are

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

then listed under the point surface names in the **Existing Points** selection list, which is located in the **Point Probes for Time Signals** group box. You can print the coordinates of the point probes to the Fluent console by selecting the desired probes and clicking the **Print Coordinates** button. Clicking the **Extract Signals** button outputs the pressure histories for the selected probes in the ASCII files; these files can then be analyzed using the Fourier Transform Dialog Box (p. 2602) (available from the Plots Task Page (p. 2593)) by selecting **Process File Data** and then reading the exported ASCII file using the **Load Input File...** button. The exported files names are by default the point probe names with the extension `.dat`. The **File Name Root** field allows you to specify a prefix for these file names.

The **FFT Surface Variables** tab is shown in Figure 22.13: The FFT Surface Variables Tab of the Acoustic Source FFT Dialog Box for the Octave Bands (p. 1227) and Figure 22.14: The FFT Surface Variables Tab of the Acoustic Source FFT Dialog Box for a Set of Individual Modes (p. 1228). This tab allows you to **Create** new Fluent variables using the computed spectrum fields, which reside in the storage area and correspondingly increase its size. These variables will be defined only on those wall face zones for which the Fourier transformation has been computed; on the other face zones, they will show zero values. Created variables can be used for any kind of further postprocessing in Fluent, which typically includes contour plotting (see the variables marked with the restriction *asfft* in Table 33.17: **Acoustics** Category (p. 1956)) or exporting to CFD-Post.

Figure 22.13: The FFT Surface Variables Tab of the Acoustic Source FFT Dialog Box for the Octave Bands



Variables may characterize either the individual Fourier modes (if you select **Set of Modes** from the **Modes/Frequency Bands** drop-down list), or frequency bands that each represent a range of frequencies (if you select the other choices in the **Modes/Frequency Bands** drop-down list). The three types of frequency bands that are available include:

- **Octave Bands**

  These are proportional bands corresponding to the standard technical octaves.

- **1/3 Octave Bands**

  These are proportional bands corresponding to the standard technical thirds.

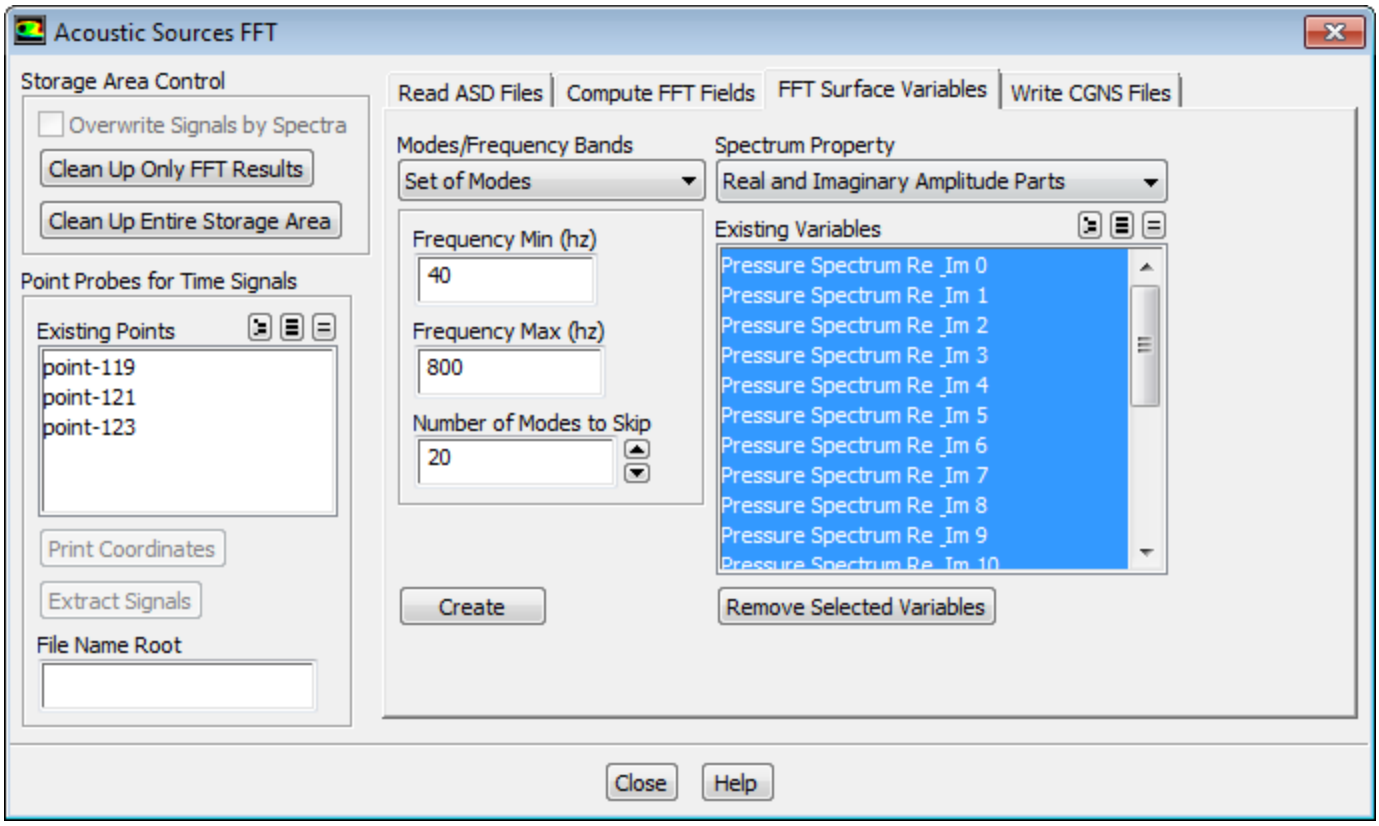
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Constant Width Bands**

  These are user-defined equidistant bands.

The **Spectrum Property** drop-down list displays the type of variables created according to the selected choice in the **Modes/Frequency Bands** drop-down list. For a set of modes, the created variables are the real and the imaginary parts of the complex Fourier amplitudes, with a pair of variables per specified mode. For the frequency bands, the created variables are the surface pressure level (SPL) fields, in decibels. The transformation to the decibel units is done by default using the standard acoustic reference pressure value of 2×10^{-5} Pa; you can change this value before creating the variables by opening the **Acoustic Models** dialog box, selecting the **Ffowcs-Williams & Hawkings** model, and revising the **Reference Acoustic Pressure**.

Figure 22.14: The FFT Surface Variables Tab of the Acoustic Source FFT Dialog Box for a Set of Individual Modes



The allowed number of variables makes it possible for you to simultaneously keep SPL for all octave and 1/3 octave bands, as well as SPL for up to 20 constant width bands and the real and imaginary amplitudes for up to 20 individual frequencies. The SPL variables for the octaves and the 1/3 octaves include the band central frequency in their names (for example, **SPL for Octave Band at 250Hz (dB)** or **SPL for 1/3-Octave Band at 1.25kHz (dB)**). As for the user-defined frequencies and constant width bands, their variables are simply numbered from 0 to 19. In order to provide you with the information about the meaning of each such variable, Fluent prints a table in the console when you click **Create**. The following is an example of the table corresponding to the settings in the previous figure:

```
Creating variables for 20 modes from 40.5983 Hz to 466.88 Hz every 22.4359 Hz.
f( 0) = 40.5983 Hz     f( 1) = 63.0342 Hz
f( 2) = 85.4701 Hz     f( 3) = 107.906 Hz
f( 4) = 130.342 Hz     f( 5) = 152.778 Hz
f( 6) = 175.214 Hz     f( 7) = 197.65 Hz
f( 8) = 220.085 Hz     f( 9) = 242.521 Hz
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
f(10) = 264.957 Hz     f(11) = 287.393 Hz
f(12) = 309.829 Hz     f(13) = 332.265 Hz
f(14) = 354.701 Hz     f(15) = 377.137 Hz
f(16) = 399.573 Hz     f(17) = 422.009 Hz
f(18) = 444.444 Hz     f(19) = 466.88 Hz

Variables for frequencies above 466.88 Hz are not created,
because the limit of 20 modes has been reached.
```

You can analyze more than 20 individual modes or constant width bands, if you process them by portions of 20 items and delete the processed variables by selecting them in the **Existing Variables** list and clicking the **Remove Selected Variables** button.

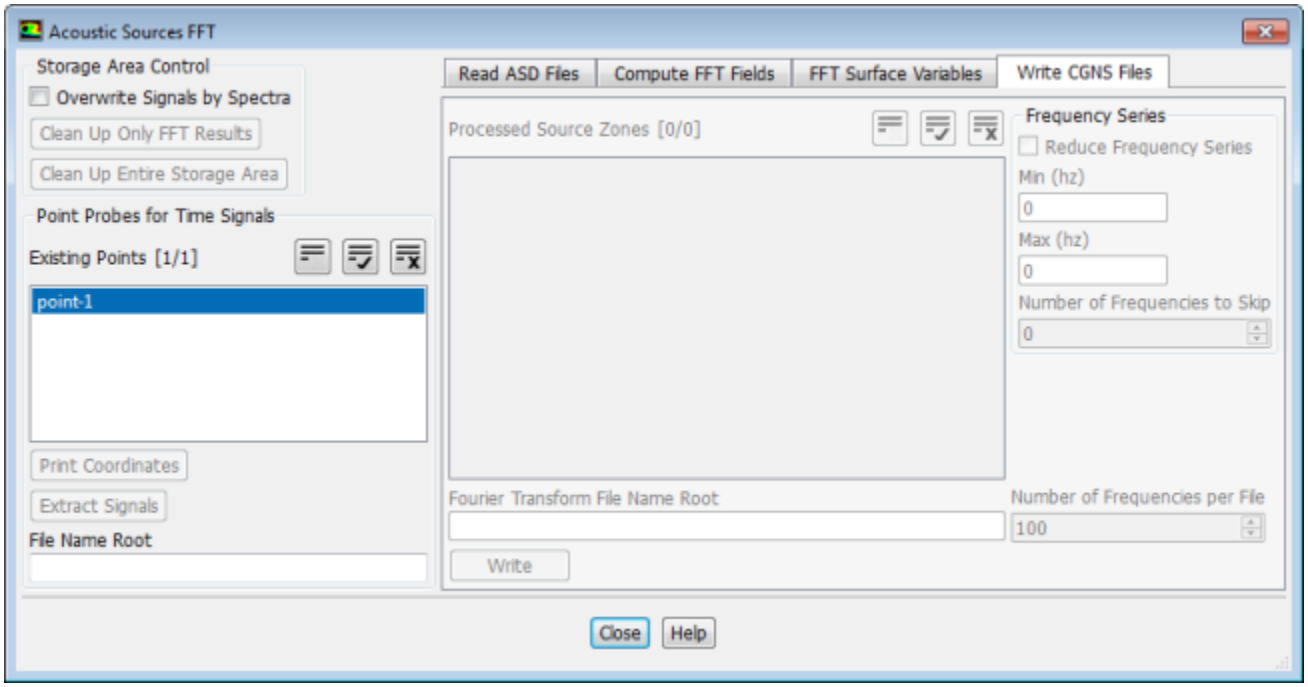
Clicking either the **Clean Up Entire Storage Area** or the **Clean Up Only FFT Results** button on the left side of the dialog box not only de-allocates the spectrum storage array, but also removes all created variables.

The **Write CGNS Files** tab is shown in Figure 22.15: The Write CGNS Files Tab of the Acoustic Source FFT Dialog Box (p. 1230). This tab allows you to specify the zones you want to export to CGNS, by selecting them in the **Processed Source Zones** list. You can reduce the exported spectrum data by enabling the **Reduce Frequency Series** option and specifying the frequency range (**Min** and **Max** frequencies), as well as the frequency step (the **Number of Frequencies to Skip** integer field). The latter must be used with care and only if really needed, because the Fourier amplitudes in the broadband noise spectrum depend on the frequency resolution. By skipping frequencies from the exported spectrum (for example, exporting every other Fourier mode), you will artificially coarsen the frequency resolution. However, the exported amplitudes will not be automatically re-scaled for the increased frequency step, but will retain their originally computed values.

Pressure spectra, which result from the scale-resolving flow simulations, may contain thousands of frequencies. Together with dense surface meshes, this can result in a very large amount of data to export. The total size of the disk space required to store fields of the complete spectra is approximately equal to the total size of the ASD files containing the wall pressure histories. Therefore, the wall pressure spectra are written to a series of CGNS files according to the **Number of Frequencies per File** field. It is recommended that you select an appropriate value for this parameter, to avoid extremely large output files or a large number of very small files. The total number of frequencies in the spectrum can be seen in the **Number of Modes** field of the **Compute FFT Fields** tab (see Figure 22.12: The Compute FFT Fields Tab of the Acoustic Source FFT Dialog Box (p. 1226)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 22.15: The Write CGNS Files Tab of the Acoustic Source FFT Dialog Box



Finally, you can specify a name root for the exported CGNS files in the **Fourier Transform File Name Root** field. For example, if you enter `wall_pressure_spectrum` and then click the **Write** button, Fluent will create the following set of files in your working folder:

wall_pressure_spectrum.cgns

wall_pressure_spectrum_1.cgns

wall_pressure_spectrum_2.cgns

...

wall_pressure_spectrum.flst

If you leave the name root field empty, the files will be named **cgns**, **1.cgns**, **2.cgns**, **...**, **flst**. The first CGNS file (without a number in its name) contains the mesh data, as well as links to all of the other CGNS files, which in turn contain the spectrum fields with the specified number of modes per file. The CGNS links work like the file links in the UNIX file system: in order to import spectra to the ANSYS Mechanical software, you only have to specify the name of the first file, and all other files will be imported automatically if they reside in the same folder. The file with the extension **flst** is an ASCII file, which contains the list of the exported frequencies.

Important

Note the following:

- You should not specify a relative path as a part of the **Fourier Transform File Name Root** input. If you do that, all files will be exported in the desired folder according to the specified path. However, the CGNS links, which are written in the first CGNS file, will also include the specified path. Therefore, the linked CGNS files will become readable only after the correspondent file re-location—for example, moving the first CGNS file in your Fluent working folder.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The public domain CGNS library, which is used to process CGNS files in Fluent and ANSYS Mechanical, has a performance issue when run on Windows. Namely, for CGNS files that have many links (as is the case in this application), the input / output operations may be very slow if the CGNS files are located on a network disk. Therefore, for Windows it is strongly recommended that you select either a built-in local disk or a USB disk directly connected to your computer as the location of the CGNS files. This warning concerns both the export of CGNS files from Fluent and their import in ANSYS Mechanical.

22.3. Using the Broadband Noise Source Models

In this section, the procedure for setting up and using the broadband noise source models is outlined first, followed by descriptions of each of the steps involved.

The general procedure for carrying out a broadband noise source calculation in ANSYS Fluent is as follows:

1. Calculate a steady or unsteady RANS solution.

2. Enable the broadband noise model and set the associated model parameters.

   Setup → **Models** → **Acoustics** **Edit...**

3. Postprocess the noise sources.

   Results → **Graphics** → **Contours** **Edit...**

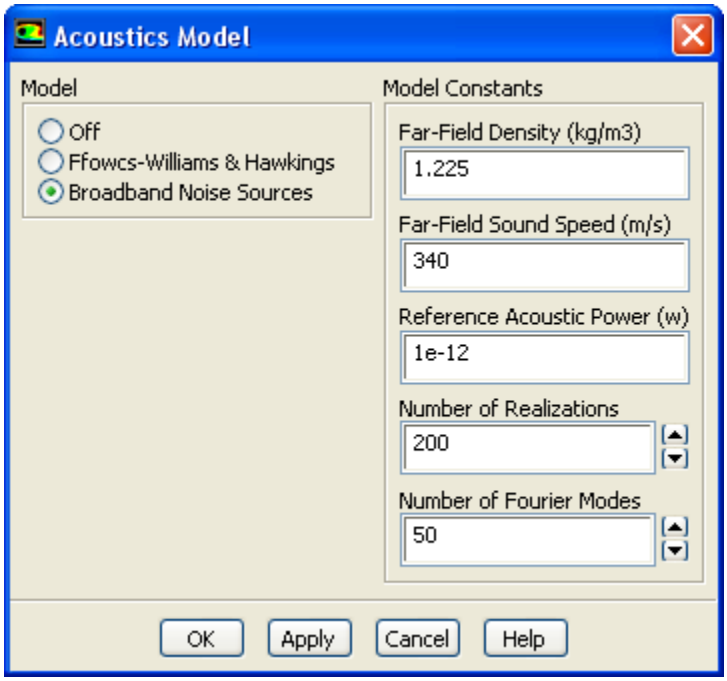
For additional information, see the following sections:
22.3.1. Enabling the Broadband Noise Source Models
22.3.2. Postprocessing the Broadband Noise Source Model Data

22.3.1. Enabling the Broadband Noise Source Models

To enable the broadband noise sources models, select **Broadband Noise Sources** in the **Acoustics Model** dialog box (Figure 22.16: The Acoustics Model Dialog Box for Broadband Noise (p. 1232)).

Setup → **Models** → **Acoustics** **Edit...**

Figure 22.16: The Acoustics Model Dialog Box for Broadband Noise



22.3.1.1. Setting Model Constants

Under **Model Constants** in the **Acoustics Model** dialog box, specify the relevant acoustic parameters and constants used by the model. See Enabling the FW-H Acoustics Model (p. 1208) for the definitions of **Far-Field Density** and **Far-Field Sound Speed**.

Reference Acoustic Power
(for example, P_{ref} in Equation 15.14 in the Theory Guide) is used to compute the acoustic power outputs in decibels (dB). The default value is 10^{-12}. Note that the units for the reference acoustic power will be different in 2D (W/m^2) and 3D (W/m^3) cases.

Number of Realizations
is the number of samples used in the SNGR to compute the averaged source terms of LEE and Lilley's equations. The default value is 200.

Number of Fourier Modes
(N in Equation 15.34 in the Theory Guide) is the number of the Fourier modes used to compute the turbulent velocity field and its derivatives. The turbulent velocity field is then used to compute the LEE and Lilley's source terms. The default value is 50.

22.3.2. Postprocessing the Broadband Noise Source Model Data

The final step in the broadband noise source modeling process is the postprocessing of acoustic power and noise source data. The following variables are available in the **Acoustics...** postprocessing category:

- **Acoustic Power Level (dB)**
- **Acoustic Power**
- **Jet Acoustic Power Level (dB)** (axisymmetric models only)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Jet Acoustic Power** (axisymmetric models only)
- **Surface Acoustic Power Level (dB)**
- **Surface Acoustic Power**
- **Lilley's Self-Noise Source**
- **Lilley's Shear-Noise Source**
- **Lilley's Total Noise Source**
- **LEE Self-Noise X-Source**
- **LEE Shear-Noise X-Source**
- **LEE Total Noise X-Source**
- **LEE Self-Noise Y-Source**
- **LEE Shear-Noise Y-Source**
- **LEE Total Noise Y-Source**
- **LEE Self-Noise Z-Source** (3D models only)
- **LEE Shear-Noise Z-Source** (3D models only)
- **LEE Total Noise Z-Source** (3D models only)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 23: Modeling Discrete Phase

This chapter describes how to use the Lagrangian discrete phase capabilities available in ANSYS Fluent. For information about the theory behind the discrete phase models, see Discrete Phase in the Theory Guide. Information is organized into the following sections:

23.1. Introduction
23.2. Steps for Using the Discrete Phase Models
23.3. Setting Initial Conditions for the Discrete Phase
23.4. Setting Boundary Conditions for the Discrete Phase
23.5. Setting Material Properties for the Discrete Phase
23.6. Solution Strategies for the Discrete Phase
23.7. Postprocessing for the Discrete Phase
23.8. Parallel Processing for the Discrete Phase Model

23.1. Introduction

In addition to solving transport equations for the continuous phase, ANSYS Fluent allows you to simulate a discrete second phase in a Lagrangian frame of reference. This second phase consists of spherical particles (which may be taken to represent droplets or bubbles) dispersed in the continuous phase. ANSYS Fluent computes the trajectories of these discrete phase entities, as well as heat and mass transfer to/from them. The coupling between the phases and its impact on both the discrete phase trajectories and the continuous phase flow can be included.

ANSYS Fluent provides the following discrete phase modeling options:

- calculation of the discrete phase trajectory using a Lagrangian formulation that includes the discrete phase inertia, hydrodynamic drag, and the force of gravity, for both steady and unsteady flows
- prediction of the effects of turbulence on the dispersion of particles due to turbulent eddies present in the continuous phase
- heating/cooling of the discrete phase
- vaporization and boiling of liquid droplets
- combusting particles, including volatile evolution and char combustion to simulate coal combustion
- optional coupling of the continuous phase flow field prediction to the discrete phase calculations
- droplet breakup and coalescence
- consideration of particle/particle collisions and voidage of discrete phase

These modeling capabilities allow ANSYS Fluent to simulate a wide range of discrete phase problems including particle separation and classification, spray drying, aerosol dispersion, bubble stirring of liquids, liquid fuel combustion, and coal combustion. The physical equations used for these discrete phase calculations are described in Discrete Phase in the *Fluent Theory Guide*, and instructions for setup, solution, and postprocessing are provided in the remaining sections of this chapter.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Alternative models for multiphase systems use the Euler-Euler approach rather than the Euler-Lagrange approach used in the Discrete Phase Model. The Euler-Euler models are discussed in Modeling Multiphase Flows (p. 1365) along with the Dense Discrete Phase Model which is a hybrid Euler-Euler and Euler-Lagrange approach.

For additional information, see the following sections:

23.1.1. Concepts
23.1.2. Limitations

23.1.1. Concepts

This section introduces several concepts in the treatment of discrete phase particles in Fluent that are important to understand in order to get the most out of the remaining information in this chapter.

23.1.1.1. Uncoupled vs. Coupled DPM
23.1.1.2. Steady vs. Unsteady Tracking
23.1.1.3. Parcels

23.1.1.1. Uncoupled vs. Coupled DPM

When the fluid changes the particles, there will be corresponding effects on the fluid. For example, when drag force acts on a particle, the exchange of momentum can change the fluid flow. When simulating particles using DPM, you can choose whether or not to include these effects in the flow solution; these alternatives are called Coupled and Uncoupled DPM. See Options for Interaction with the Continuous Phase (p. 1241) for details about how to specify whether your simulation uses Coupled or Uncoupled DPM.

In Uncoupled DPM, the only purpose of the DPM particles is for postprocessing, and so particles are not tracked except when you request them, for example to calculate and display particle tracks. The particles can still change by heat and mass transfer, but the corresponding changes (such as vapor from an evaporating droplet) do not affect the flow solution.

In a Coupled DPM simulation, the effects of the particles are used to influence the flow solution. These effects are transmitted to the flow as DPM Sources. The DPM solution and the flow solution should reach a converged, self-consistent solution. Therefore, there are several options for running these solutions together—see Solution Strategies for the Discrete Phase (p. 1325).

23.1.1.2. Steady vs. Unsteady Tracking

As described in Steps for Using the Discrete Phase Models (p. 1240), to set up a DPM simulation you specify the starting conditions of a set of particles by defining an injection. By specifying boundary conditions and physical sub-models, you also specify how these particles interact with other zones in the geometry and how they eventually leave the model—for instance, they might bounce off some walls but be trapped by others.

You must also specify how Fluent is to track the particles you have defined. If Steady Tracking is enabled, then, as soon as a particle is released, it is tracked until it reaches its final destination according to the specified boundary behavior (or until a fixed number of particle time steps have been used). Therefore, each particle typically travels through many cells of the model, interacting with the flow and (in a Coupled DPM simulation) changing the DPM Sources in each cell. These sources influence the flow solution for a defined number of iterations or time steps—the flow solution can be steady or unsteady. Then, if required, a new set of particle trajectories is tracked, the DPM Sources are updated, and the sequence is repeated. An example of using Steady DPM with unsteady flow is when the chosen flow models require a transient simulation, although the final goal is a steady solution.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If Unsteady Tracking is enabled, then each particle is advanced by a specified number of particle time steps, not necessarily reaching a final destination, before the flow solution is updated. When Unsteady DPM is coupled to unsteady flow solution, the particles and the flow develop in time together concurrently, although different time steps can be used for DPM and flow.

Unsteady DPM can also be coupled to steady flow solution; this makes sense if there is a continuous source of DPM particles that pass through the system. For steady or unsteady flow, there are several DPM models where Unsteady Tracking is required. For example, in spray coalescence and collision models, particles change with time on the basis of interactions with other particles, so they must be tracked simultaneously

23.1.1.3. Parcels

Especially when using Coupled DPM, the mass flow rate of a particle injection will often be a required and relevant input parameter since it determines the absolute value of the DPM Sources. This mass flow rate could be converted into the number of particles injected per unit time. However, it is often prohibitive to track that number of particles in a simulation. Strictly speaking, the model therefore tracks a number of 'parcels', and each parcel is representative of a fraction of the total continuous mass flow rate (in Steady tracking) or a fraction of the total mass flow released in a time step (in Unsteady tracking).

It is still sometimes helpful to refer to each parcel as a representative particle, because it has a specified particle diameter, and its trajectory in fluid flow uses the relaxation time appropriate for a single particle. (The relaxation time is a ratio of particle momentum to drag force). However, the parcel's mass (or mass flow rate) becomes important when calculating the DPM Sources: for example, if a representative droplet loses a small amount of vapor by evaporation, the overall effect from the whole parcel will typically be much larger. Other models also use the parcel mass (or mass flow rate) to calculate total concentrations of DPM material—in particular, the Dense Discrete Phase Model uses this concentration to feed into the volume fraction of the Eulerian phase that represents the same material. See Dense Discrete Phase Model in the *Fluent Theory Guide* for details of the Dense Discrete Phase Model

The concept of parcels is particularly important in the Discrete Element Method (DEM), where parcels occupy a finite volume and obstruct other DEM parcels. The volume occupied by a parcel is calculated directly from the mass that it represents (so that a realistic density is created when parcels pack together). The equivalent 'parcel diameter' is used for calculating parcel-parcel contacts and forces. However, for trajectories through fluid, it is still the 'particle diameter' that is used. See Discrete Element Method Collision Model in the *Fluent Theory Guide* for details of the Discrete Element Model.

The number of parcels in a DPM model is chosen in the model settings, and not defined by the true number of particles. There are several inputs that can be used to adjust the number of parcels when defining initial conditions such as the number of injection locations and (for Unsteady Tracking) the injection frequency (Setting Initial Conditions for the Discrete Phase (p. 1265)) Other inputs in sub-models include: the number of sizes in a size distribution (Using the Rosin-Rammler Diameter Distribution Method (p. 1281)); the number of stochastic tries in turbulent dispersion (Specifying Turbulent Dispersion of Particles (p. 1294)); and the breakup characteristics of some sprays (Breakup (p. 1292)). A high number of parcels can be computationally expensive, but it is often helpful for convergence, so that no single parcel has an overwhelming effect on the flow In general you should arrange for enough parcels to produce a statistical sample, representative of the full range of particle behavior.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23.1.2. Limitations

23.1.2.1. Limitation on the Particle Volume Fraction

The discrete phase formulation used by ANSYS Fluent contains the assumption that the second phase is sufficiently dilute that particle-particle interactions and the effects of the particle volume fraction on the gas phase are negligible. In practice, these issues imply that the discrete phase must be present at a fairly low volume fraction, usually less than 10–12%. Note that the mass loading of the discrete phase may greatly exceed 10–12%: you may solve problems in which the mass flow of the discrete phase equals or exceeds that of the continuous phase. See Modeling Multiphase Flows (p. 1365) for information about when you might want to use one of the general multiphase models instead of the discrete phase model.

This limitation is relaxed for some variants of DPM For example, the Dense Discrete Phase Model adds effects due to friction and volume fraction, so that the concentration can approach the packing limit Where high local concentrations of spray droplets cause coalescence and collision, these phenomena can be included in some spray models Parcel-parcel contacts between solid particles are modeled in detail in Discrete Element Models, so that parcels can pack together closely.

23.1.2.2. Limitation on Modeling Continuous Suspensions of Particles

The steady-particle Lagrangian discrete phase model is suited for flows in which particle streams are injected into a continuous phase flow with a well-defined entrance and exit condition. The Lagrangian model does not effectively model flows in which particles are suspended indefinitely in the continuum, as occurs in solid suspensions within closed systems such as stirred tanks, mixing vessels, or fluidized beds. The unsteady-particle discrete phase model, however, is capable of modeling continuous suspensions of particles. See Modeling Multiphase Flows (p. 1365) for information about when you might want to use one of the general multiphase models instead of the discrete phase models.

23.1.2.3. Limitations on Modeling Particle Rotation

When using rotating particles, note the following limitations:

- In simulations with enabled stochastic particle collision models (Including Collision and Droplet Coalescence (p. 1252)), particle rotation is not affected by particle/particle collisions.
- Particle rotation is not available for massless particles.
- Particle rotation is not compatible with moving reference frame simulations.
- For atomizer injections, the initial angular velocity is set to zero.
- The effect of the Magnus lift force on the fluid is not taken into account in DPM coupled simulations.

23.1.2.4. Limitations on Using the Discrete Phase Model with Other ANSYS Fluent Models

The following restrictions exist on the use of other models with the discrete phase model:

- When tracking particles in parallel, the DPM model cannot be used with any of the multiphase flow models (VOF, mixture, or Eulerian—see Modeling Multiphase Flows (p. 1365)) if the shared memory option is enabled (Parallel Processing for the Discrete Phase Model (p. 1360)). (Note that using the message passing or hybrid

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

option, when running in parallel, enables the compatibility of all multiphase flow models with the DPM model.)

- When using the DPM model with the Eulerian multiphase model, the tracked particles rely only on the primary phase to compute drag, heat, and mass transfer. Also, any DPM related source terms are applied to the primary phase. Particle tracking relative to a secondary phase is not provided.
- Streamwise periodic flow (either specified mass flow rate or specified pressure drop) cannot be modeled with steady particle tracks in coupled simulation. It is possible using transient particle tracks.
- Only non-reacting particles can be included when the premixed combustion model is used.
- Surface injections will be moved with the mesh when a sliding mesh or a moving or deforming mesh is being used, however only those surfaces associated with a boundary will be recalculated. Injections from cut plane surfaces will not be moved with the mesh and will be deleted when remeshing occurs.
- The cloud model is not available for unsteady particle tracking, or in parallel, when using the message passing or hybrid option for the particles.
- The wall film model is only valid for liquid materials. If a nonliquid particle interacts with a wall film boundary, the boundary condition will default to the reflect boundary condition.
- When multiple reference frames are used in conjunction with the discrete phase model, the display of particle tracks will not, by default, be meaningful. Similarly, coupled discrete-phase calculations are not meaningful.

  An alternative approach for particle tracking and coupled discrete-phase calculations with multiple reference frames is to track particles based on absolute velocity instead of relative velocity. To make this change, use the `define/models/dpm/ options/track-in-absolute-frame` text command. Note that the results may strongly depend on the location of walls inside the multiple reference frame.

  The particle injection velocities (specified in the **Set Injection Properties** dialog box) are defined relative to the frame of reference in which the particles are tracked. By default, the injection velocities are specified relative to the local reference frame. If you enable the `track-in-absolute-frame` option, the injection velocities are specified relative to the absolute frame.
- Relative particle tracking cannot be used in combination with sliding and moving deforming meshes. If sliding and/or deforming meshes are used with the DPM model, the particles will always be tracked in the absolute frame. Switching to the relative frame is not permitted.

23.1.2.5. Limitations on Using the Hybrid Parallel Method

- The hybrid parallel DPM tracking method is not available with the following models and features:
  - the PDF Transport model
  - the DDPM (Dense DPM) model with the **Volume Fraction Approaching Packing Limit** option selected in the **Discrete Phase** dialog box
  - the **Stochastic Collision** model (with or without the droplet coalescence)
  - Graphical display of particle tracks
  - Sampling of particle tracks in sample files

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  – Export of particle tracks into particle history files

  If any of these models or features are used, Fluent will silently fall back to the **Message Passing** parallel DPM tracking option.

- The **Use DPM Domain** option of the **Hybrid** parallel DPM tracking mode will not be used in any of the following situations:

  – if the **Update DPM Sources Every Flow Iteration** option is enabled

  – if the **Contour Plots for DPM Variables** option is enabled

  – if the **Enable Node Based Averaging** option (under the **Numerics** tab) is enabled

  – if the case includes non-zonal surface injections (that is, injections that have positions defined through surfaces that are not zone surfaces)

  – if the case includes a wall / shadow pair

  – if you are not running with distributed memory on a cluster

  If the **Use DPM Domain** option is enabled in such cases, ANSYS Fluent will issue a message warning that the DPM Domain is suppressed. You can avoid this message by disabling the **Use DPM Domain** option in the **Parallel** tab of the **Discrete Phase Model** dialog box.

  For additional limitations associated with the hybrid parallel method, see Limitations on Using the Discrete Phase Model with Other ANSYS Fluent Models (p. 1238).

23.1.2.6. Limitations on Using the Lagrangian Wall Film Model

- The Lagrangian wall film model is only available with **Unsteady Particle Tracking**.

- The Lagrangian wall film model is not compatible with hanging-node mesh refinement on walls. Though the fluid meshes can be adapted, the refined meshes must not touch walls where the wall film model is enabled.

- The wall film model assumes liquid particle materials. If an inert particle interacts with a wall film boundary, it is assumed to be liquid for the wall film model calculations. For combusting particles, the wall film is applied if **Wet Combustion Model** is enabled and as long as the liquid fraction in the particle is nonzero. If a dry combusting particle interacts with a wall film boundary it will stick to the wall and follow the combusting particle laws.

- The wall film model is not compatible with the Rosseland Radiation model.

- The **Workpile Algorithm** option is not available with the wall film boundary condition. It will be disabled automatically when choosing to simulate a wall film on a wall.

23.2. Steps for Using the Discrete Phase Models

You can include a discrete phase in your ANSYS Fluent model by defining the initial position, velocity, size, and temperature of individual particles. These initial conditions, along with your inputs defining the physical properties of the discrete phase, are used to initiate trajectory and heat/mass transfer calculations. The trajectory and heat/mass transfer calculations are based on the force balance on the particle and on the convective/radiative heat and mass transfer from the particle, using the local con-

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tinuous phase conditions as the particle moves through the flow. The predicted trajectories and the associated heat and mass transfer can be viewed graphically and/or alphanumerically.

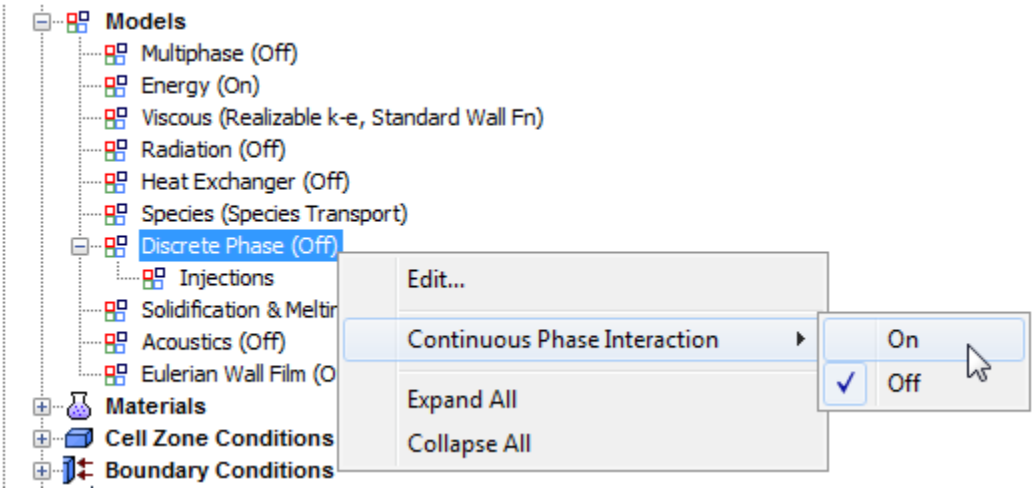
The procedure for setting up and solving a problem involving a discrete phase is outlined below, and described in detail in Options for Interaction with the Continuous Phase (p. 1241) – Postprocessing for the Discrete Phase (p. 1329). Only the steps related specifically to discrete phase modeling are shown here. For information about inputs related to other models that you are using in conjunction with the discrete phase models, see the appropriate sections for those models.

1. Enable any of the discrete phase modeling options, if relevant, as described in Options for Interaction with the Continuous Phase (p. 1241).
2. Choose a transient or steady treatment of particles as described in Steady/Transient Treatment of Particles (p. 1242).
3. Specify tracking parameters as described in Tracking Parameters for the Discrete Phase Model (p. 1246).
4. Enable the required physical submodels for the discrete phase model, as described in Physical Models for the Discrete Phase Model (p. 1249).
5. Set the numerics parameters and solve the problem, as described in Numerics of the Discrete Phase Model (p. 1260) and Solution Strategies for the Discrete Phase (p. 1325).
6. Specify the injection-specific models, initial conditions, and particle size distributions as described in Setting Initial Conditions for the Discrete Phase (p. 1265).
7. Define the boundary conditions, as described in Setting Boundary Conditions for the Discrete Phase (p. 1301).
8. Define the material properties, as described in Setting Material Properties for the Discrete Phase (p. 1313).
9. Initialize the flow field.
10. Solve the coupled or uncoupled flow (Solution Strategies for the Discrete Phase (p. 1325)).
11. For transient cases, advance the solution in time by taking the desired number of time steps. Particle positions will be updated as the solution advances in time. If you are solving an uncoupled flow, the particle position will be updated at the end of each time step. For a coupled calculation, the positions are iterated on or within each time step.
12. Examine the results, as described in Postprocessing for the Discrete Phase (p. 1329).

23.2.1. Options for Interaction with the Continuous Phase

If the discrete phase interacts (that is, exchanges mass, momentum, and/or energy) with the continuous phase, you should enable **Interaction with Continuous Phase**.

Setup → Models → Discrete Phase → Continuous Phase Interaction → On



An input for the **DPM Iteration Interval** will appear, which allows you to control the frequency at which the particles are tracked and the DPM sources are updated.

For steady-state simulations, increasing the **DPM Iteration Interval** will increase stability but require more iterations to converge.

In addition, another option exists that allows you to control the numerical treatment of the source terms and how they are applied to the continuous phase equations. For unsteady simulations, **Update DPM Sources Every Flow Iteration** is the default (and recommended) option; at every DPM Iteration, the particle source terms are recalculated. The source terms applied to the continuous phase equations transition to the new values every flow iteration based on Equation 16.440 to Equation 16.442 in the Theory Guide. This process is controlled by the under-relaxation factor, specified in the **Solution Controls** task page, see Under-Relaxation of the Interphase Exchange Terms (p. 1328).

23.2.2. Steady/Transient Treatment of Particles

The Discrete Phase Model uses a Lagrangian approach to derive the equations for the underlying physics, which are solved transiently. Transient numerical procedures in the Discrete Phase Model can be applied to resolve steady flow simulations as well as transient flows.

In the Discrete Phase Model Dialog Box (p. 2278) you have the option of choosing whether you want to treat the particles in an unsteady or a steady fashion. This option can be chosen independent of the settings for the solver. Thus, you can perform steady-state trajectory simulations even when selecting a transient solver for numerical reasons. You can also specify unsteady particle tracking when solving the steady continuous phase equations. This can be used to improve numerical stability for very large particle source terms or simply for postprocessing purposes. Whenever you enable a breakup or collision model to simulate sprays, the **Unsteady Particle Tracking** will be switched on automatically.

When **Unsteady Particle Tracking** is enabled, several new options appear. If steady-state equations are solved for the continuous phase, you simply enter the **Particle Time Step Size** and the **Number of Time Steps**, thus tracking particles every time a DPM iteration is conducted. When you increase the **Number of Time Steps**, the droplets penetrate the domain faster.

When solving unsteady equations for the continuous phase, you must decide whether you want to use **Fluid Flow Time Step** to inject the particles, or you prefer a **Particle Time Step Size** independent of the **Fluid Flow Time Step**. With the latter option, you can use the Discrete Phase Model in combination with changes in the time step for the continuous equations, as it is done when using adaptive flow time stepping.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you do not use **Fluid Flow Time Step**, you must decide when to inject the particles for a new time step. You can either **Inject Particles at Particle Time Step** or at the **Fluid Flow Time Step**. In any case, the particles will always be tracked in such a way that they coincide with the flow time of the continuous flow solver, as long as the maximum number of time steps used to compute a single trajectory is sufficient (see Tracking Parameters for the Discrete Phase Model (p. 1246) for details).

You can use a user-defined function (`DEFINE_DPM_TIMESTEP`) to change the time step for DPM particle tracking. The time step can be prescribed for special applications where a certain time step is needed. For more information about changing the time step size for DPM particle tracking, see `DEFINE_DPM_TIMESTEP` in the Fluent Customization Manual.

Important

When the density-based solver is used with the explicit unsteady formulation, the particles are advanced once per time step and are calculated at the start of the time step (before the flow is updated).

Additional inputs are required for each injection in the Set Injection Properties Dialog Box (p. 2796), as detailed in Defining Injection Properties (p. 1286). For **Unsteady Particle Tracking**, the injection **Start Time** and **Stop Time** must be specified under **Point Properties**. Injections with start and stop times set to zero will be injected only at the start of the calculation ($t=0$). If the In-Cylinder mesh motion is enabled, the start and stop times are replaced by **Start Crank Angle** and **Stop Crank Angle**, respectively. The injection specified in this way will be repeated at the starting and stopping crank angle if the simulation is run through more than one cycle. Changing injection settings during a transient simulation will not affect particles currently released in the domain. At any point during a simulation, you can clear particles that are currently in the domain by clicking the **Clear Particles** button in the Discrete Phase Model Dialog Box (p. 2278).

You can choose the **Parcel Release Method** to determine how ANSYS Fluent creates parcels. For an overview of the concept of parcels, see Parcels (p. 1237). The methods available are:

standard

injects a single parcel per injection stream per time step. The number of particles in the parcel, NP, is determined as follows:

$$NP=\dot{m}_s\frac{\Delta t}{m_p} \tag{23.1}$$

where,

NP is the number of particles in a parcel

$\dot{m}_s$ is the mass flow rate of the particle stream

Δt is the time step

m_p is the particle mass

This is the default method.

constant-number

injects parcels with a user-specified number of particles per parcel. The number of parcels is determined to satisfy the specified mass-flow rate and particle size distribution for the injection.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

constant-mass
injects parcels with a user-specified parcel mass. The number of parcels is determined to satisfy the specified mass-flow rate and particle size distribution for the injection.

constant-diameter
injects parcels with a user-specified parcel diameter. The number of parcels is determined to satisfy the specified mass-flow rate and particle size distribution for the injection.

Note that for atomizer injections, **file** injections, or injections that use the `DEFINE_DPM_INJECTION_INIT` user defined function, the Fluent solver automatically uses the default standard parcel release method. Other parcel release methods are not available.

For cases involving sprays and particle size distributions in general, the recommended setting for **Parcel Release Method** is **constant-number**. For DEM simulations, you can use **constant-diameter** or **constant-mass** to ensure that the parcel diameter does not exceed the size of the smallest cells in the computational mesh.

Note that a lower value specified for **constant-number**, **constant-mass**, or **constant-diameter** will result in a larger number of parcels injected and a finer discretization of the DPM phase. This may be beneficial for accuracy and stability of the calculation, at the expense of additional computational cost.

The **Parcel Release Method** is specified in the **Parcel** tab of the **Set Injection Properties** dialog box.

You can also choose one of several methods to control when the particles are advanced.

- If the **DPM Iteration Interval** is less than the number of iterations specified to converge the continuous phase between time steps, then sub-iterations are done. Here, particles are tracked to their new positions during a time step and DPM sources are updated; particles are then returned to their original state at the beginning of the time step. At the end of the time step, particles are advanced to their new positions based on the continuous-phase solution.
- If the **DPM Iteration Interval** is larger than the number of iterations specified to converge the continuous phase between time steps, the particles are advanced at the beginning of the time step to compute the particle source terms.
- If you specify a value of zero as the **DPM Iteration Interval**, the particles are advanced at the end of the time step. Note that if you have previously enabled the resetting of DPM source terms at the beginning of every time step using the TUI command `define/models/dpm/interaction/reset-sources-at-timestep?`, ANSYS Fluent will disable this option once you set the **DPM Iteration Interval** to zero, because this feature combination is not advisable.

In all the above cases, you must provide a sufficient number of particle source term updates to better control when the particles are advanced, see Figure 23.36: Effect of Number of Source Term Updates on Source Term Applied to Flow Equations (p. 1329).

Important

- In steady-state discrete phase modeling, particles do not interact with each other and are tracked one at a time in the domain.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If the collision model is used, you will not be able to set the **DPM Iteration Interval**. Refer to Collision and Droplet Coalescence Model Theory in the *Fluent Theory Guide* for details about this limitation.

Note

As explained in Setting Initial Conditions for the Discrete Phase (p. 1265), the injection flow rate is defined per unit meter of depth in 2D problems. In such cases, each injected parcel is considered to be a solid cylinder of diameter d_{2D} and depth 1 meter. The parcel volume is then expressed as:

$$V_{2D}=\frac{\pi}{4}{d_{2D}}^2\cdot 1\,m$$

However, the particles that are dense-packed in the parcel are assumed to be spheres rather than pucks (ice-hockey). The volume of each particle is the same as in 3D cases:

$$V_p=\frac{\pi}{6}{d_p}^3$$

Consider next the theoretical atomic packing factor β of spherical particles in a cylinder of one meter depth, also expressed as the volume ratio of all particles to all injected parcels.

In 2D, the particles of diameter d_p are stacked on top of one another in a cylinder of diameter d_p and depth 1 meter. The total number of particles that can be stacked in the cylinder in such a way is $1\,\mathrm{m}/d_p$. The total volume occupied by these particles is

$$V_1=\frac{\pi}{6}{d_p}^3\cdot 1\,\mathrm{m}/d_p$$

The volume of the surrounding parcel cylinder of depth 1 m is

$$V_2=\frac{\pi}{4}{d_p}^2\cdot\ 1\,\mathrm{m}$$

The atomic packing factor β will have a constant value of 2/3 when expressed as:

$$\beta=V_1/V_2=\left(\frac{\pi}{6}{d_p}^3\cdot 1\,\mathrm{m}/d_p\right)\Big/\left(\frac{\pi}{4}{d_p}^2\cdot\ 1\,\mathrm{m}\right)=\frac{2}{3}$$

In order to use the same parameters (spring constant k and coefficient of restitution η) for the DEM spring dashpot models in both 3D and 2D simulations, parcels of the same mass must be injected in 3D and 2D.

In 3D, the injected parcel is a sphere with diameter d_{3D} and mass $m_{3D}=\rho_p\cdot\frac{\pi}{6}{d_{3D}}^3$. The number of spherical particles in a spherical parcel is calculated by

$$NP=m_{3D}/m_p=\rho_p\cdot\frac{\pi}{6}{d_{3D}}^3\Big/\rho_p\cdot\frac{\pi}{6}{d_p}^3$$

In 2D, the parcel is a cylinder of 1 m depth. This cylinder has the same mass as the spherical parcel in 3D ($m_{2D}=m_{3D}$), but its volume differs and is calculated as:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$V_{prcl,2D}=\frac{\pi}{4}{d_{2D}}^2\cdot\ 1\,\mathrm{m}$$

The number of particles per cylindrical parcel NP and the volume they occupy is the same as in 3D, that is

$$V_{p,2D}=V_{p,3D}=NP\ \frac{\pi}{6}{d_p}^3$$

Since the ratio of these two volumes must be equal to the theoretical atomic packing factor β, we obtain

$$\left(NP\cdot\ \frac{\pi}{6}{d_p}^3\right)\Big/\left(\frac{\pi}{4}{d_{2D}}^2\cdot\ 1m\right)=\ \beta=\frac{2}{3}$$

The parcel diameter in 2D can then be calculated as:

$$d_{2D}=\sqrt{NP\cdot\ {d_p}^3/1\,\mathrm{m}}=\sqrt{{d_{3D}}^3/1\,\mathrm{m}}$$

Note that in 2D, the parcel diameter d_{2D} is used as a collision range and it is different than the parcel diameter d_{3D} in 3D.

For each parcel release method, the input values for the parcel mass and parcel diameter (m_{3D} and d_{3D}, respectively) are the same in both 3D and 2D, while the output values for parcel diameters are different: d_{2D} in 2D simulations and d_{3D} in 3D simulations.

23.2.3. Tracking Parameters for the Discrete Phase Model

You will use two parameters to control the time integration of the particle trajectory equations:

- the maximum number of time steps

  This factor is used to abort trajectory calculations when the particle never exits the flow domain.

- the length scale/step length factor

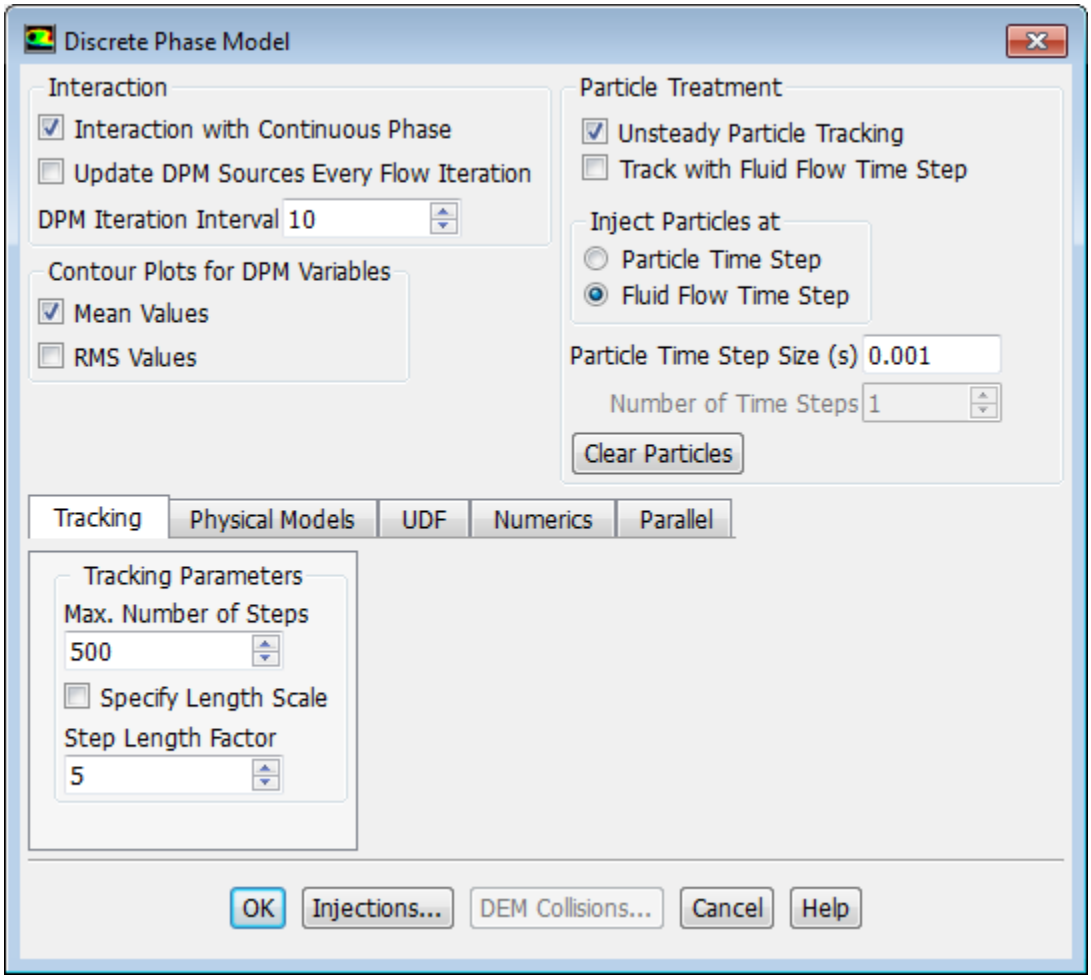
  This factor is used to set the time step for integration within each control volume.

Each of these parameters is set in the Discrete Phase Model Dialog Box (p. 2278) (Figure 23.1: The Discrete Phase Model Dialog Box and the Tracking Parameters (p. 1247)) under **Tracking Parameters** in the **Tracking** tab.

Setup → **Models** → **Discrete Phase** → **Edit...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.1: The Discrete Phase Model Dialog Box and the Tracking Parameters



Max. Number of Steps

is the maximum number of time steps used to compute a single particle trajectory via integration of Equation 16.1 in the Theory Guide. When the maximum number of steps is exceeded, ANSYS Fluent abandons the trajectory calculation for the current particle injection and reports the trajectory fate as "incomplete". The limit on the number of integration time steps eliminates the possibility of a particle being caught in a recirculating region of the continuous phase flow field and being tracked infinitely. The default value for this parameter is 50,000 for steady-state particle tracking and 500 for unsteady particle tracking. Note that you may easily create problems in which the default value is insufficient for completion of the trajectory calculation. In this case, when trajectories are reported as incomplete within the domain and the particles are not recirculating indefinitely, you can increase the maximum number of steps (up to a limit of 10^9).

Note

If, when opening a previous simulation with unsteady DPM particle tracking, you receive a warning reporting that a fraction of the discrete phase mass is "behind in time", you can use the reported percentage values to determine if it is a significant enough discrepancy to warrant revising the simulation to have a higher value for **Max. Number of Steps**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Length Scale

controls the integration time step size used to integrate the equations of motion for the particle. The integration time step is computed by ANSYS Fluent based on a specified length scale L, and the velocity of the particle (u_p) and of the continuous phase (u_c):

$$\Delta t = \frac{L}{u_p + u_c} \tag{23.2}$$

where L is the **Length Scale** that you define. As defined by Equation 23.2 (p. 1248), L is proportional to the integration time step and is equivalent to the distance that the particle will travel before its motion equations are solved again and its trajectory is updated. A smaller value for the **Length Scale** increases the accuracy of the trajectory and heat/mass transfer calculations for the discrete phase.

(Note that particle positions are always computed when particles enter/leave a cell; even if you specify a very large length scale, the time step used for integration will be such that the cell is traversed in one step.)

Length Scale will appear in the **Discrete Phase Model** dialog box when the **Specify Length Scale** option is enabled.

Step Length Factor

also controls the time step size used to integrate the equations of motion for the particle. It differs from the **Length Scale** in that it allows ANSYS Fluent to compute the time step in terms of the number of time steps required for a particle to traverse a computational cell. To set this parameter instead of the **Length Scale**, turn off the **Specify Length Scale** option.

The integration time step is computed by ANSYS Fluent based on a characteristic time that is related to an estimate of the time required for the particle to traverse the current continuous phase control volume. If this estimated transit time is defined as Δt^*, ANSYS Fluent chooses a time step Δt as

$$\Delta t = \frac{\Delta t^*}{\lambda} \tag{23.3}$$

where λ is the **Step Length Factor**. As defined by Equation 23.3 (p. 1248), λ is inversely proportional to the integration time step and is roughly equivalent to the number of time steps required to traverse the current continuous phase control volume. A larger value for the **Step Length Factor** decreases the discrete phase integration time step. The default value for the **Step Length Factor** is 5. **Step Length Factor** will appear in the **Discrete Phase Model** dialog box when the **Specify Length Scale** option is off (the default setting).

For steady-state particle tracking, one simple general rule to follow when setting the parameters above is that if you want the particles to advance through a domain consisting of N mesh cells into the main flow direction, the **Step Length Factor** times N should be approximately equal to the **Max. Number of Steps**.

When **Accuracy Control** is enabled in the **Numerics** tab, the settings for **Step Length Factor** and **Length Scale** will be used only to estimate the time step of the first integration step. In all subsequent integration steps, the particle integration time step is adapted to achieve the tolerance specified in Numerics of the Discrete Phase Model (p. 1260).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23.2.4. Drag Laws

There are eight drag laws for the particles that can be selected for each injection. These are specified on the **Physical Models** tab of the **Set Injection Properties** dialog box. See Specifying Injection-Specific Physical Models (p. 1290) for details.

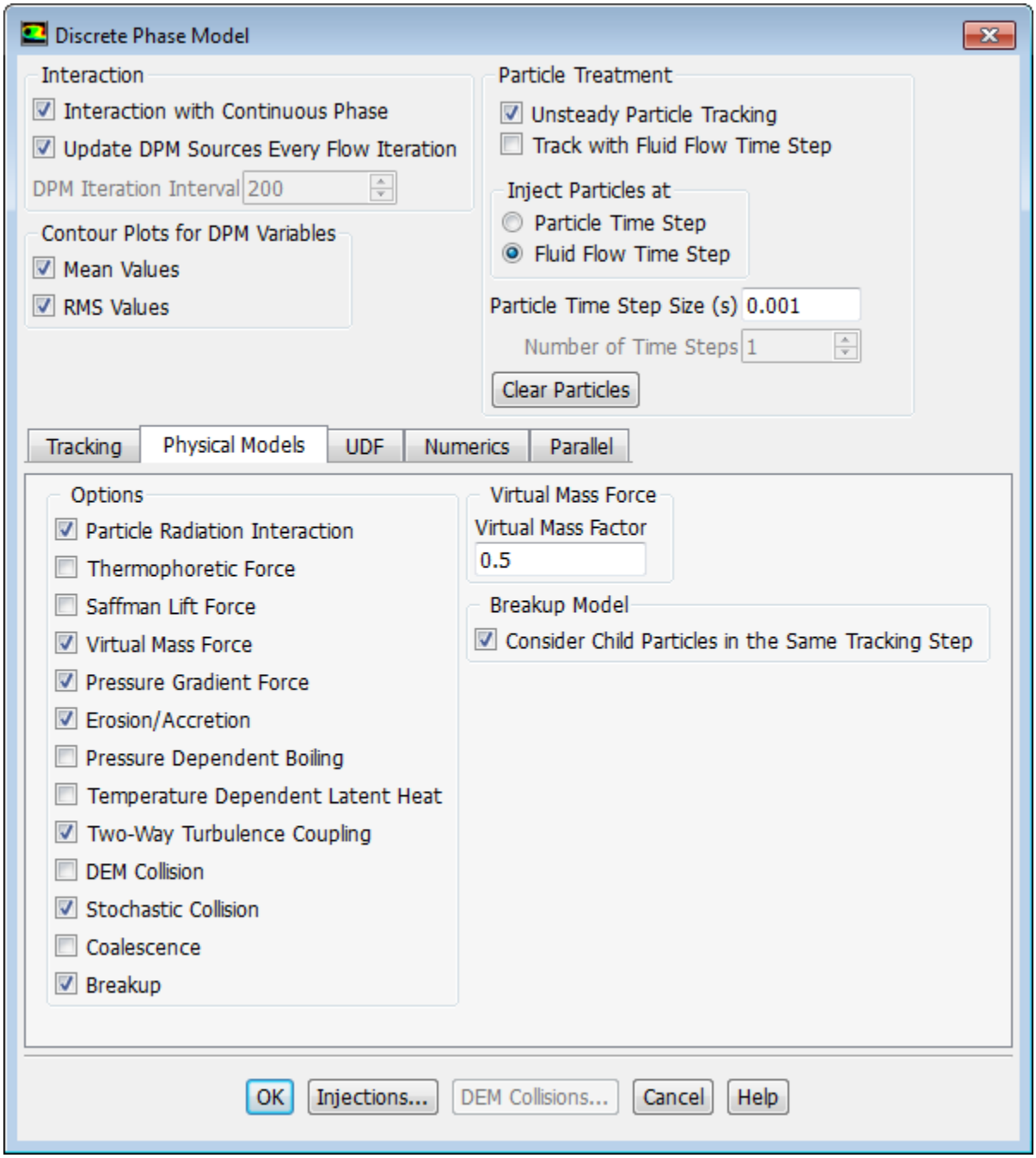
23.2.5. Physical Models for the Discrete Phase Model

This section provides instructions for using the optional discrete phase models available in ANSYS Fluent. All of them can be enabled in the **Physical Models** tab of the **Discrete Phase Model** dialog box (Figure 23.2: The Discrete Phase Model Dialog Box and the Physical Models (p. 1250)).

Setup → **Models** → **Discrete Phase** **Edit...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.2: The Discrete Phase Model Dialog Box and the Physical Models



23.2.5.1. Including Radiation Heat Transfer Effects on the Particles

If you want to include the effect of radiation heat transfer to the particles (Equation 5.34 in the Theory Guide), you must enable the **Particle Radiation Interaction** option under the **Physical Models** tab, in the Discrete Phase Model Dialog Box (p. 2278) (Figure 23.2: The Discrete Phase Model Dialog Box and the Physical Models (p. 1250)). You also must define additional properties for the particle materials (emissivity and scattering factor), as described in Description of the Properties (p. 1317). This option is available only when the P-1 or discrete ordinates radiation model is used.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23.2.5.2. Including Thermophoretic Force Effects on the Particles

If you want to include the effect of the thermophoretic force on the particle trajectories (Equation 16.11 in the Theory Guide), enable the **Thermophoretic Force** option under the **Physical Models** tab, in the Discrete Phase Model Dialog Box (p. 2278). You also must define the thermophoretic coefficient for the particle material, as described in Description of the Properties (p. 1317).

23.2.5.3. Including Saffman Lift Force Effects on the Particles

For sub-micron particles, you can also model the lift due to shear (the Saffman lift force, described in Saffman's Lift Force in the Theory Guide) in the particle trajectory. To do this, enable the **Saffman Lift Force** option under the **Physical Models** tab, in the Discrete Phase Model Dialog Box (p. 2278).

23.2.5.4. Including the Virtual Mass Force and Pressure Gradient Effects on Particles

In cases where the density of the fluid approaches or exceeds the density of the particles (for example liquid flow with gaseous bubbles), it is recommended that you include the Virtual Mass and Pressure Gradient forces in the particle force balance. To do this, enable **Virtual Mass Force** and **Pressure Gradient Force** under the **Physical Models** tab in the Discrete Phase Model Dialog Box (p. 2278). See Other Forces in Fluent Theory Guide for information on the Virtual Mass and Pressure Gradient forces.

23.2.5.5. Monitoring Erosion/Accretion of Particles at Walls

To monitor particle erosion and accretion rates at wall boundaries:

1. Enable **Erosion/Accretion** in the **Physical Models** tab of the Discrete Phase Model Dialog Box (p. 2278).

   Enabling the **Erosion/Accretion** option will cause the erosion and accretion rates to be calculated at wall boundary faces when particle tracks are updated.

2. For each wall zone, select erosion models and specify erosion model parameters as described in Setting Particle Erosion and Accretion Parameters (p. 1309).

23.2.5.6. Enabling Pressure Dependent Boiling

With this option you can modify the condition for switching from droplet vaporization (Law 2) to boiling (Law 3). By default ANSYS Fluent will switch from the vaporization to the boiling law when the particle temperature has reached the boiling point defined for the droplet material (Equation 16.118 in the Theory Guide). When the **Pressure Dependent Boiling** option is enabled, the switching condition will change to $P_{sat}>P$, where P_{sat} is the saturation vapor pressure at the droplet temperature and P is the domain pressure. If $P_{sat}<P$ while in the boiling law, the model will switch back to the vaporization law. If this option is enabled it is essential to enter the appropriate droplet saturation vapor pressure data to cover the complete pressure/temperature range in your model.

When **Pressure Dependent Boiling** is enabled, then the **Temperature Dependent Latent Heat** model automatically applies (see Including the Effect of Droplet Temperature on Latent Heat (p. 1252)).

Setting the **Pressure Dependent Boiling** option has no effect on the multicomponent particles, where switching from the vaporization to the boiling regime is always based on the component saturation vapor pressures (see Multicomponent Particle Definition (Law 7) in the Theory Guide) Finally, selection of the **Pressure Dependent Boiling** option is not available with the real-gas models, as the pressure dependence always applies. See Using the Cubic Equation of State Models with the Lagrangian Dispersed Phase Models (p. 548) for more information.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For pressures higher than the critical pressure of the droplet material ($P>P_c$), a boiling point T_b, defined as the temperature at the saturation vapor pressure ($P=P_{sat}$), cannot be determined as the vapor pressure curve is defined only up to the critical point. Under supercritical pressure conditions ($P>P_c$), the vaporization models are applicable at droplet temperatures below the critical point ($T<T_c$), and the switching condition from vaporization to boiling is never met. For supercritical pressure conditions, you must enter the droplet saturation vapor pressure data for the complete pressure/temperature range up to the critical point T_c.

When **Pressure Dependent Boiling** is enabled, you can use the text user interface command

```
/define/models/dpm/options/allow-supercritical-pressure-vaporization?
```

to enforce the switching from vaporization to boiling even if the boiling point T_b is not calculated from the vapor pressure data.

If the pressure in your model is above critical ($P>P_c$), you must retain the default setting (`yes`) for `allow-supercritical-pressure-vaporization?`.

For subcritical pressure conditions ($P<P_c$), the setting for `allow-supercritical-pressure-vaporization?` will have no effect unless the boiling point calculation fails, which usually indicates that either the vapor pressure data do not cover the complete temperature/pressure range for your model, or inaccurate data have been entered. In that case, if `allow-supercritical-pressure-vaporization?` is set to `no`, the normal boiling point T_b defined for the droplet material will be imposed as a switching condition from vaporization to boiling.

23.2.5.7. Including the Effect of Droplet Temperature on Latent Heat

To include the droplet temperature effects on the latent heat as described in Equation 16.111 in the Theory Guide, enable **Temperature Dependent Latent Heat** under the **Physical Models** tab. If you enable this option you must provide accurate temperature-dependent specific heat data for both the droplet and the evaporating species materials.

23.2.5.8. Including the Effect of Particles on Turbulent Quantities

Particles can damp or produce turbulent eddies [76] (p. 2891). In ANSYS Fluent, the formulation described in [29] (p. 2888) and [6] (p. 2887) is used to compute turbulence modulation. Damping occurs when the particle diameter is less than one tenth of the turbulent length scale. For larger particle diameters turbulence kinetic energy is produced [37] (p. 2888).

If you want to consider these effects in the chosen turbulence model, you can enable this using **Two-Way Turbulence Coupling**, under the **Physical Models** tab.

23.2.5.9. Including Collision and Droplet Coalescence

To include the effect of collisions, as described in Collision and Droplet Coalescence Model Theory in the Theory Guide, select **Stochastic Collision** and **Coalescence** under **Options**.

Note

Coalescence will appear under **Options** after **Stochastic Collision** has been enabled.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23.2.5.10. Including the DEM Collision Model

The DEM collision model is suitable for simulating granular matter, where such simulations are characterized by a high volume fraction of particles, and the particle-particle interaction is important. See Modeling Collision Using the DEM Model (p. 1253) for details about using this model.

23.2.5.11. Including Droplet Breakup

To model droplet breakup in ANSYS Fluent, enable **Breakup** under **Options**. By default, this will enable breakup for all suitable injections. You can then specify the breakup models and parameters (or disable breakup) for each individual injection on the **Physical Models** tab in the **Set Injection Properties** dialog box as described in Breakup (p. 1292).

When **Breakup** is enabled, you can collect new generated child droplets and track them within the same time step by enabling **Consider Child Particles in the Same Tracking Step**. This option provides higher accuracy and can lead to faster convergence within a transient flow time step.

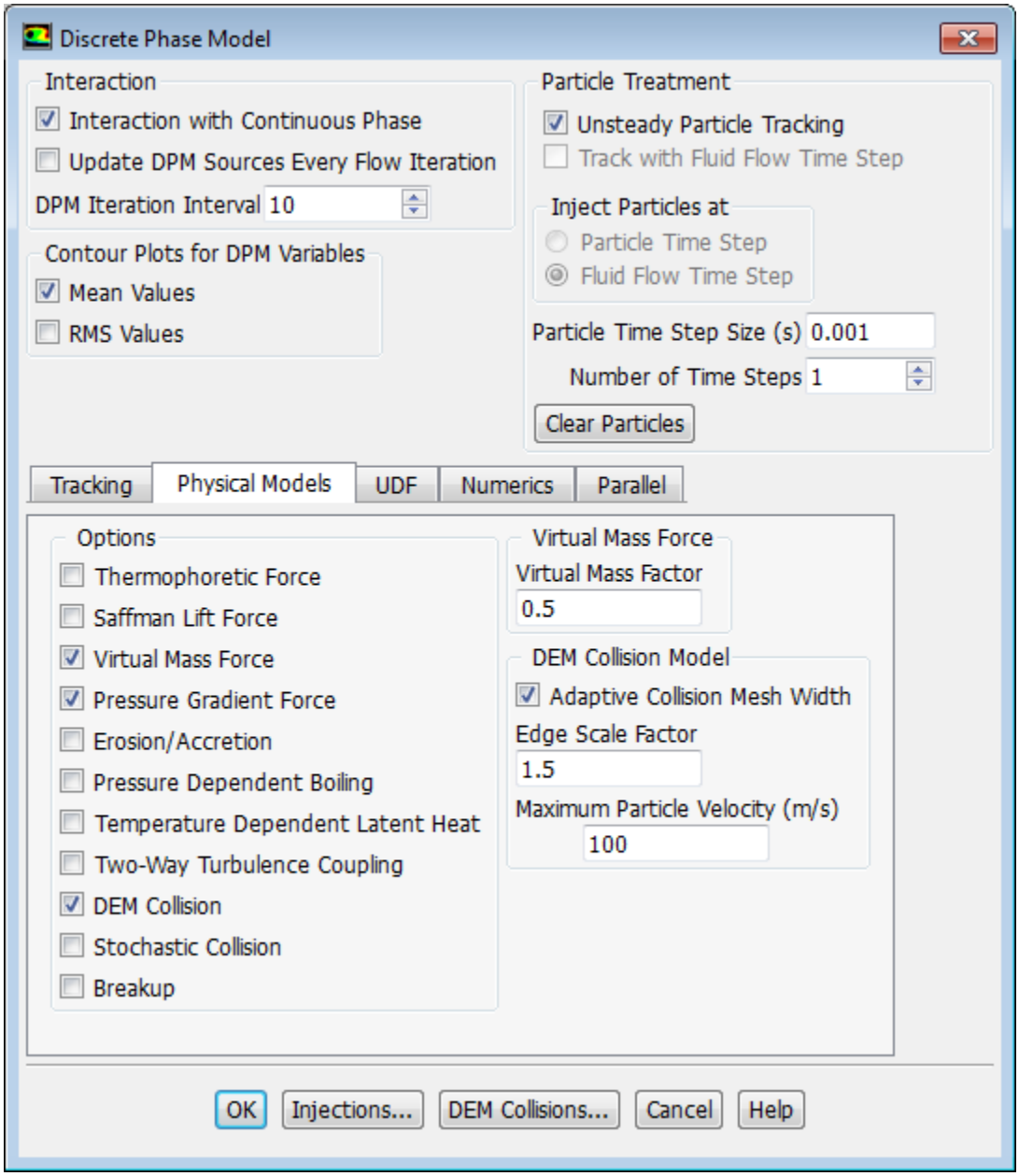
23.2.5.12. Modeling Collision Using the DEM Model

To enable the DEM collision model, select **DEM Collision** under **Options**. A detailed description of this model can be found in Discrete Element Method Collision Model in the Theory Guide.

1. In the **Physical Models** tab of the **Discrete Phase Model** dialog box

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.3: Discrete Phase Dialog Box with DEM Collision Model



a. Specify the settings your model requires under **DEM Collision Model**.

b. (Optional) By default, **Adaptive Collision Mesh Width** is enabled. This adjusts the width of the collision mesh to the largest parcel diameter multiplied by the **Edge Scale Factor**. If **Adaptive Collision Mesh Width** is disabled, a fixed **Static Collision Mesh Width** has to be given in the chosen units of length.

c. The **Maximum Particle Velocity** limits the maximum particle velocity to a physically plausible range.

2. Set up the injection.

Setup → Models → Discrete Phase → Injections New...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

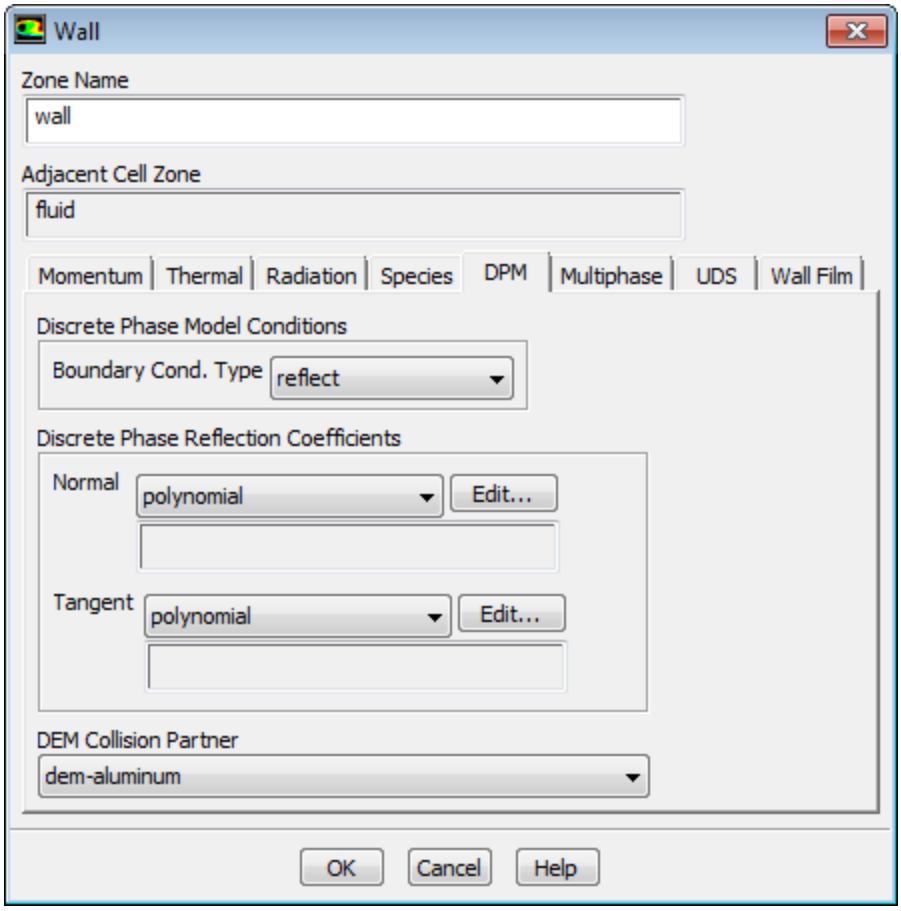
a. In the **Set Injection Properties** dialog box, select a name from the **DEM Collision Partner** drop-down list that will serve as the collision partner. A default name using the particle material will be suggested.

> **Note**
>
> Selecting none from the drop-down list indicates that the particles released from this injection do not participate in the DEM collision computation.

b. If you want to use the rolling friction collision law (see step 4), under the **Physical Models** tab, enable **Particle Rotation** for those injections whose particles take part in collisions with rolling friction.

3. Set the boundary conditions for the discrete phase as you normally would. If the **Boundary Cond. Type** is **reflect**, select a name from the **DEM Collision Partner** drop-down list to designate the collision partner For example, in Figure 23.4: Wall Boundary Condition for the DEM Model (p. 1255), a wall boundary condition will suggest a wall material name, which will designate the collision partner. The name will have a **dem-** prefix However, if the **DEM Collision Partner** is **none**, the wall will reflect particles like any other DPM particles using the settings for the **Discrete Phase Reflection Coefficients** in the **Normal** and **Tangent** directions. These settings generally apply for particles that are not colliding according to DEM, such as massless particles.

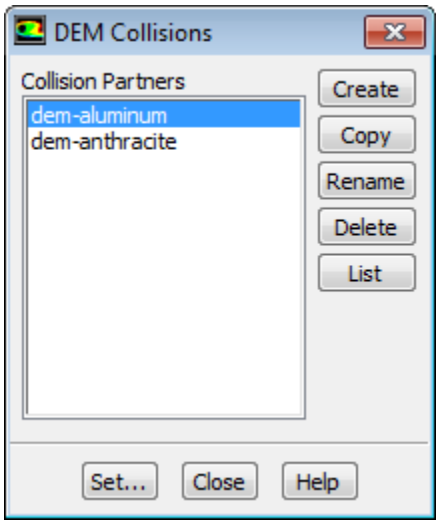
Figure 23.4: Wall Boundary Condition for the DEM Model



4. Define the particle interaction between the pairs of the collision partners.

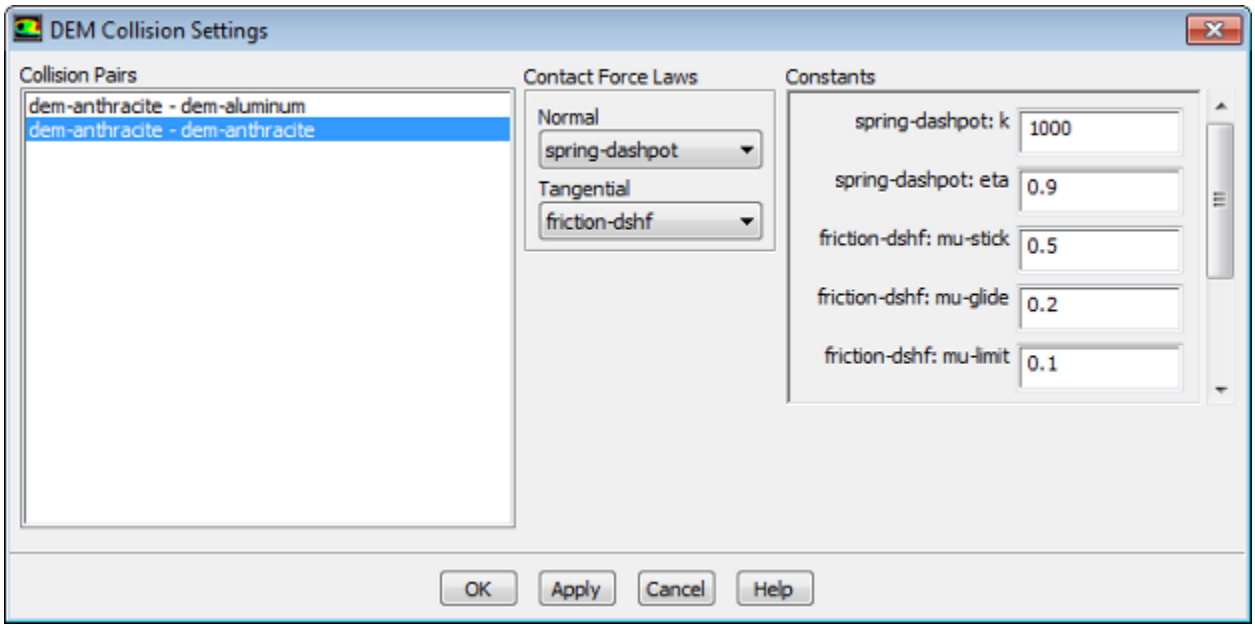
a. Click the **DEM Collisions...** button at the bottom of the **Discrete Phase Model** dialog box. The **DEM Collisions** dialog box (Figure 23.5: Collision Dialog Box (p. 1256)) will appear where you can manage the collision partners. You can **Create**, **Copy**, **Rename**, **Delete**, and **List** collision partners. To define collision laws for a collision partner, select a collision partner from the **Collision Partners** list and click the **Set...** button, or simply double-click the collision partner in the list.

Figure 23.5: Collision Dialog Box



b. In the **DEM Collision Settings** dialog box (Figure 23.6: DEM Collision Settings Dialog Box (p. 1256)), a list of all the possible **Collision Pairs** will exist.

Figure 23.6: DEM Collision Settings Dialog Box



i. Select a pair of collision partners from the **Collision Pairs** list.

ii. For this pair, select the **Contact Force Laws** that best describe the collision between these two partners.

The **Normal** contact force laws that exist are:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **spring**
- **spring-dashpot**
- **hertzian**
- **hertzian-dashpot**

You can also choose to exclude the contact force law by selecting **none**.

For the **Tangential** contact force law (if considered), you can select form the following friction collision laws:
- **friction-dshf**
- **friction-dshf-rolling** (an extension of the **friction-dshf** friction collision law including rolling friction)

Each of the laws is described in the following sections:

- The Spring Collision Law in the *Fluent Theory Guide*
- The Spring-Dashpot Collision Law in the *Fluent Theory Guide*
- The Hertzian Collision Law in the *Fluent Theory Guide*
- The Hertzian-Dashpot Collision Law in the *Fluent Theory Guide*
- The Friction Collision Law in the *Fluent Theory Guide*
- Rolling Friction Collision Law for DEM in the *Fluent Theory Guide*

iii. Enter the **Constants** for the chosen **Contact Force Laws**. Refer to Theory in the Theory Guide for guidance on how to find reasonable values of the force-law constants.

iv. Repeat for all other collision pairs.

v. Click **OK** or **Apply** to apply these settings.

Note

- The choice of collision laws does not depend on the order in the pair of collision partners.
- Failing to specify a force law for a collision pair implies that respective particles will not collide.
- Rolling friction will only be factored in for a particular DEM particle injection if the **Particle Rotation** option is selected in the **Set Injection Properties** dialog box (**Physical Properties** tab).

Note

For parallel calculations, the DEM collision model can be optimized by enabling the hybrid optimization method for DPM. This method balances the load across machines, and, within each machine, the hybrid parallel DPM method is used to make sure the load is balanced by multi-threading. For more information, see the description of the **Hybrid Optimization** option in the **Weighting** tab discussion in Partitioning (p. 2033).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23.2.5.12.1. Limitations

The following limitations currently apply when using the DEM collision model:

- Axisymmetric geometry cannot be used.
- Dynamic mesh motion cannot be used.
- The Eulerian wall film model cannot be used.
- Shear stress boundary conditions cannot be used at walls.
- Marangoni stress boundary condition cannot be used at walls.
- DEM particles do not transfer heat from particle to particle during contact.

23.2.5.12.2. Numeric Recommendations

To better preserve energy during particle collision, the following settings in the **Numerics** tab are recommended and will be automatically set when DEM Collisions are enabled.

1. Disable **Accuracy Control** to ensure that DEM time stepping remains small.
2. Disable **Automated** under **Tracking Scheme Selection**.
3. Select **implicit** from the **Tracking Scheme** drop-down list.

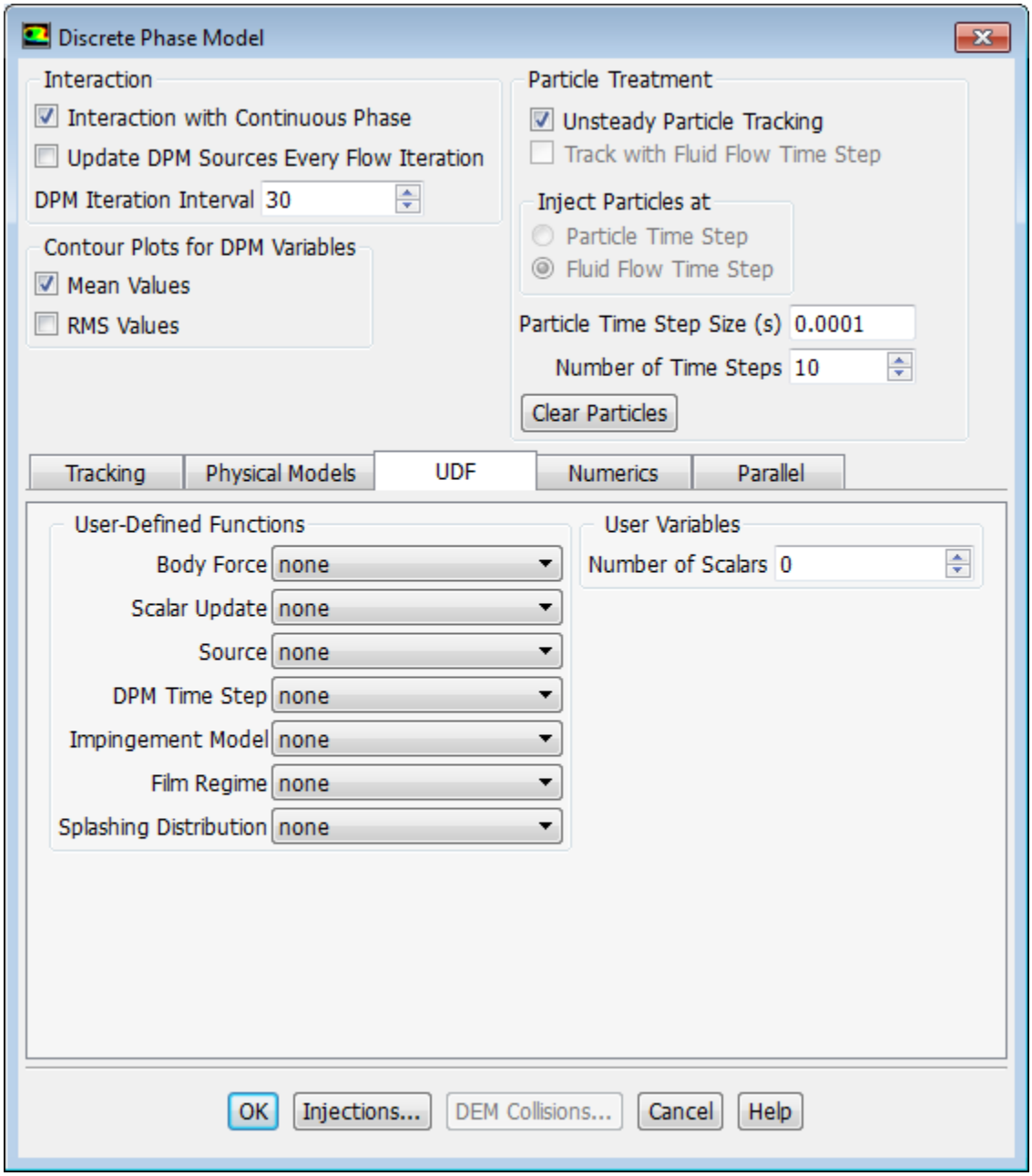
Refer to Numerics for Tracking of the Particles (p. 1261) for more information about the settings in the **Numerics** tab.

23.2.6. User-Defined Functions

User-defined functions can be used to customize the discrete phase model to include additional body forces, modify interphase exchange terms (sources), calculate or integrate scalar values along the particle trajectory, and incorporate nonstandard erosion rate definitions. More information about user-defined functions can be found in the Fluent Customization Manual.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.7: The Discrete Phase Model Dialog Box and the UDFs



In the Discrete Phase Model Dialog Box (p. 2278), under **User-Defined Functions** in the **UDF** tab, there are drop-down lists labeled **Body Force**, **Scalar Update**, **Source**, **Spray Collide Function**, and **DPM Time Step** (Figure 23.7: The Discrete Phase Model Dialog Box and the UDFs (p. 1259)). If **Erosion/Accretion** is enabled under the **Physical Models** tab, there will be an additional drop-down list labeled **Erosion/Accretion**. The **Impingement Model**, **Film Regime** and **Splashing Distribution** drop-down lists will become available if either **wall-film** is selected for **Boundary Cond. Type** in the **DPM** tab of the **Wall** dialog box or the **Eulerian Wall Film** model is enabled. Note that user-defined Impingement Model and Splashing Distribution will overwrite the impingement/splashing model selected for the Lagrangian or Eulerian wall film boundary condition.

These lists will show available user-defined functions that can be selected to customize the discrete phase model (see Fluent Customization Manual for more information).

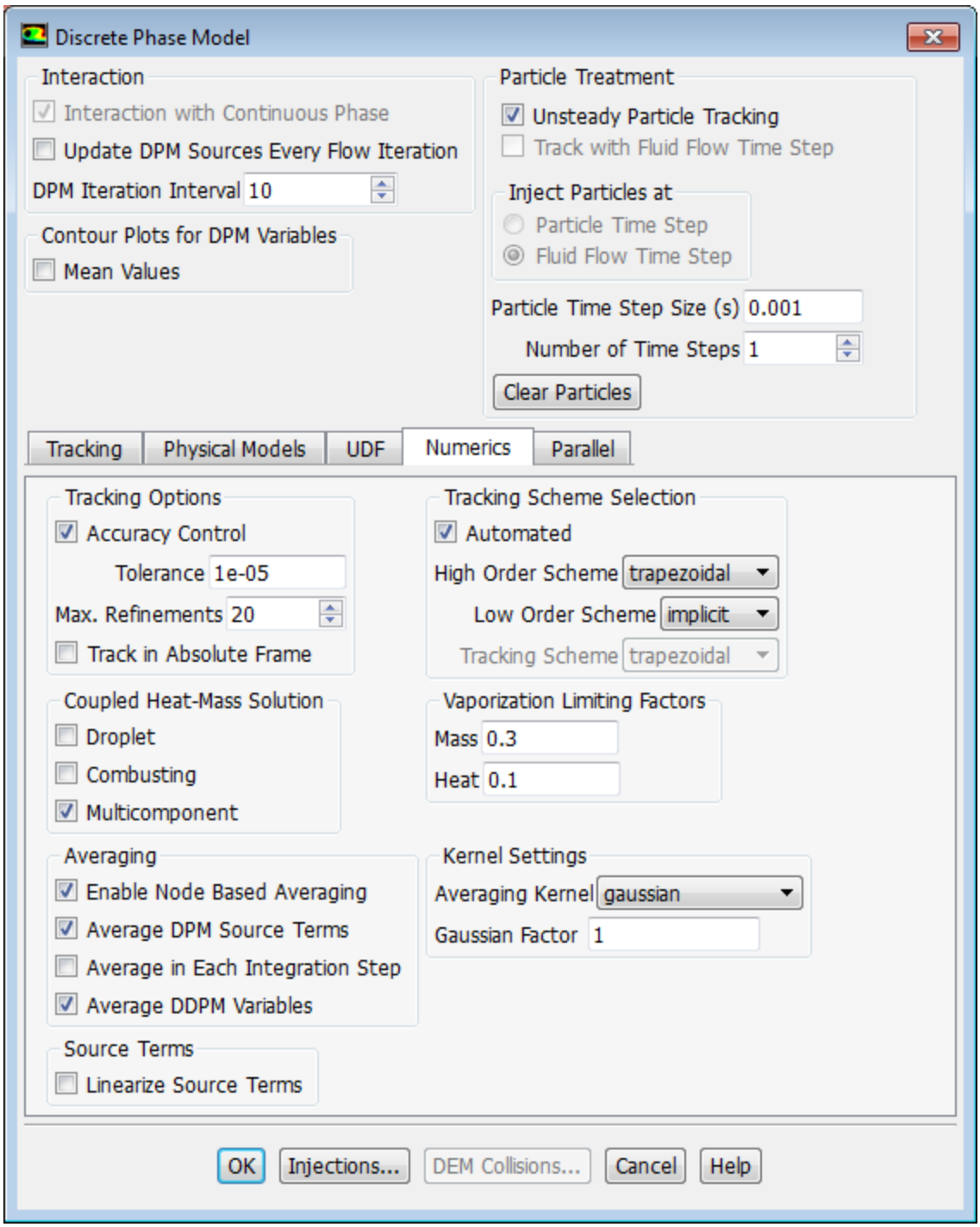
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In addition, you can specify a **Number of Scalars** which are allocated to each particle and can be used to store information when implementing your own particle models.

23.2.7. Numerics of the Discrete Phase Model

The underlying physics of the Discrete Phase Model is described by ordinary differential equations (ODE) as opposed to the continuous flow which is expressed in the form of partial differential equations (PDE). Therefore, the Discrete Phase Model uses its own numerical mechanisms and discretization schemes, which are completely different from other numerics used in ANSYS Fluent.

Figure 23.8: The Discrete Phase Model Dialog Box and the Numerics



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Numerics** tab gives you control over the numerical schemes for particle tracking as well as solutions of heat and mass equations (Figure 23.8: The Discrete Phase Model Dialog Box and the Numerics (p. 1260)).

23.2.7.1. Numerics for Tracking of the Particles

To solve equations of motion for the particles, the following numerical schemes are available:

implicit
uses an implicit Euler integration of Equation 16.1 in the Theory Guide which is unconditionally stable for all particle relaxation times.

trapezoidal
uses a semi-implicit trapezoidal integration.

analytic
uses an analytical integration of Equation 16.1 of the Theory Guide where the forces are held constant during the integration.

runge-kutta
facilitates a 5th order Runge Kutta scheme derived by Cash and Karp [19] (p. 2887).

For additional details, see Solution Strategies for the Discrete Phase (p. 1325).

You can either choose a single tracking scheme, or switch between higher order and lower order tracking schemes using an automated selection based on the accuracy to be achieved and the stability range of each scheme. In addition, you can control how accurately the equations need to be solved.

Accuracy Control
enables the solution of equations of motion within a specified tolerance. This is done by computing the error of the integration step and reducing the integration step if the error is too large. If the error is within the given tolerance, the integration step will also be increased in the next steps.

Tolerance
is the maximum relative error that has to be achieved by the tracking procedure. Based on the numerical scheme, different methods are used to estimate the relative error. The implemented Runge-Kutta scheme uses an embedded error control mechanism. The error of the other schemes is computed by comparing the result of the integration step with the outcome of a two step procedure with half the step size.

Max. Refinements
is the maximum number of step size refinements in one single integration step. If this number is exceeded the integration will be conducted with the last refined integration step size.

Automated Tracking Scheme Selection
provides a mechanism to switch in an automated fashion between numerically stable lower order schemes and higher order schemes, which are stable only in a limited range. In situations where the particle is far from hydrodynamic equilibrium, an accurate solution can be achieved very quickly with a higher order scheme, since these schemes need less step refinements for a certain tolerance. When the particle reaches hydrodynamic equilibrium, the higher order schemes become inefficient since their step length is limited

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

to a stable range. In this case, the mechanism switches to a stable lower order scheme and facilitates larger integration steps.

Important

This mechanism is only available when **Accuracy Control** is enabled.

Higher Order Scheme
can be chosen from the group consisting of **trapezoidal** and **runge-kutta** scheme.

Lower Order Scheme
consists of **implicit** and the exponential **analytic** integration scheme.

Tracking Scheme
is selectable only if **Automated** is switched off. You can choose any of the tracking schemes. You also can combine each of the tracking schemes with **Accuracy Control**.

Note

- When **DEM Collisions** are enabled, ANSYS Fluent will automatically set **Tracking Scheme** to **implicit** and disable **Accuracy Control**.
- The equation for rotating particle motion (Equation 16.210 in the *Fluent Theory Guide*) is solved using the Euler implicit discretization scheme.

23.2.7.2. Including Coupled Heat-Mass Solution Effects on the Particles

By default, the particle heat and mass equations are solved in a segregated manner using an implicit Euler integration over the time step used for the trajectory calculation If you enable the **Coupled Heat-Mass Solution** option for the **Droplet**, **Combusting**, or **Multicomponent** particles, ANSYS Fluent will solve the corresponding equations using a coupled ODE solver with error tolerance control. The increased accuracy, however, comes at the expense of increased computational time.

To ensure solution accuracy for the **Droplet** and **Multicomponent** particles, ANSYS Fluent will automatically switch to the coupled algorithm during the integration time step when the evaporated mass is greater than the limiting mass change Δm_p (defined in Equation 23.4 (p. 1262)), or when the particle temperature change is greater than the limiting temperature change ΔT_p (defined in Equation 23.5 (p. 1262)).

$$\Delta m_p = m_p a_m \tag{23.4}$$

where m_p is the particle mass (kg) and a_m is the vaporization limiting factor for the mass.

$$\Delta T_p = min\left(\left|T_p - T_\infty\right| a_T, T_p a_T\right) \tag{23.5}$$

where T_p is the particle temperature (K), T_∞ is the bulk temperature (K), and a_T is the vaporization limiting factor for the temperature.

You will enter the **Vaporization Limiting Factors** for **Mass** a_m and **Heat** a_T. The defaults 0.3 and 0.1 are recommended.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23.2.7.3. Tracking in a Reference Frame

Particle tracking is related to a coordinate system. With **Track in Absolute Frame** enabled, you can choose to track the particles in the absolute reference frame. All particle coordinates and velocities are then computed in this frame. The forces due to friction with the continuous phase are transformed to this frame automatically.

In rotating flows it might be appropriate for numerical reasons to track the particles in the relative reference frame. If several reference frames exist in one simulation, then the particle velocities are transformed to each reference frame when they enter the fluid zone associated with this reference frame.

When the impact of particles with walls in multiple rotating reference frames is important, as it is the case with a rotating impeller in a stationary baffled tank, it is necessary to model the flow as a sliding mesh simulation.

23.2.7.4. Node Based Averaging of Particle Data

In general, the presence of DPM parcels may affect the continuous phase fluid flow meaning that discrete phase source terms and particle data such as fluid drag, temperature, volume fraction, and velocity can be integrated into the flow solver. By default in ANSYS Fluent, the effects of a particular DPM parcel are applied only to the cell containing that parcel. As an alternative, **Node Based Averaging** is offered which distributes the effects of a DPM parcel among neighboring nodes in the mesh. This allows a reduction of grid dependency of DPM simulations, since each parcel's effects on the flow solver are distributed more smoothly across neighboring cells.

You can enable **Node Based Averaging** in the **Numerics** tab of the **Discrete Phase Model** dialog box, as shown in Figure 23.8: The Discrete Phase Model Dialog Box and the Numerics (p. 1260).

When enabled, **Node Based Averaging** is applied to particle velocity, particle temperature, particle volume fraction, and DPM concentration. By default, **Average DPM Source Terms** is also enabled, which applies node averaging to the source terms as well. When the Dense Discrete Phase Model is being used, the **Average DDPM Variables** option appears to provide node based averaging for the discrete phase specific variables, such as DDPM particle drag. When **Contour Plots for DPM Variables** is enabled (Reporting of Discrete Phase Variables (p. 1351)), these quantities will also be node averaged.

Average in Each Integration Step should be enabled if you find that the flux report is showing discrepancies for mass flow rates of particles injected into and leaving the domain. This option increases computation time so it is not enabled by default.

Once you have enabled **Node Based Averaging**, you must also select the **Averaging Kernel** to accumulate the particle related data on the mesh nodes and to redistribute them back to the finite volume cells. The kernel and related settings are specified in the **Kernel Settings** section of the **Discrete Phase Model** dialog box. The following kernels are available:

- **nodes-per-cell**
- **shortest-distance**
- **inverse-distance**
- **gaussian**

  If you use the **gaussian** kernel, you must also specify the **Gaussian Factor**, which determines the width of the Gaussian distribution. The default value is 1.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For details about the averaging and kernel equations, refer to Node Based Averaging in the Theory Guide.

23.2.7.5. Linearized Source Terms

Source terms for discrete phase momentum, energy, and species can be linearized with respect to the cell variable, ϕ:

$$S_{DPM,\phi}=S_{const}+S_{lin}\phi \tag{23.6}$$

This linearization strongly increases numerical stability for steady flows. For transient flows, it typically allows the use of larger time steps and larger under-relaxation factors for the DPM model.

You can enable source term linearization with the **Linearize Source Terms** option in the **Numerics** tab of the **Discrete Phase Model** dialog box.

The source term linearization can be combined with the **Average DPM Source Terms** (Node Based Averaging of Particle Data (p. 1263)) and **Update DPM Sources Every Flow Iteration** (Options for Interaction with the Continuous Phase (p. 1241)) options. However, you should disable **Average DPM Source Terms** when using linearized DPM source terms for vaporizing particles. Combination of these two features in vaporization cases may lead to numerical instabilities and unphysical results for the gas temperature.

23.2.7.6. Staggering of Particles in Space and Time

In order to obtain a better representation of an injector, the particles can be *staggered* spatially and/or temporally. When particles are staggered spatially, ANSYS Fluent randomly samples from the region in which the spray is specified (for example, the sheet thickness in the pressure-swirl atomizer) so that as the calculation progresses, trajectories will originate from the entire region. This allows the entire geometry specified in the atomizer to be sampled while specifying fewer streams in the input dialog box, thus decreasing computational expense.

When spatial staggering is enabled for non-atomizer injections, a stagger radius must be specified to define the region from which particles are released. The stagger radius is specified in the text user interface (TUI), using the command:

```
/define/models/dpm/options/stagger-radius
```

You can also enable injection-specific spatial staggering of the particles by selecting the **Stagger Positions** option and specifying **Stagger Radius** in the **Point Properties** tab of the **Set Injection Properties** dialog box. For more information, see the appropriate sections for injection-specific point properties.

When injecting particles in a transient calculation using relatively large time steps in relation to the spray event, the particles can clump together in discrete bunches. The clumps do not look physically realistic, though ANSYS Fluent calculates the trajectory for each particle as it passes through a cell and the coupling to the gas phase is properly accounted for. To obtain a statistically smoother representation of the spray, the particles can be staggered in time. During the first time step, the particle is tracked for a random percentage of its initial step. This results in a sample of the initial volume swept out by the particle during the first time step and a smoother, more uniform spatial distribution at longer time intervals.

The setup commands for staggering are available in the text user interface (TUI), under

- `stagger-spatially-atomizer-injections?`
- `stagger-spatially-standard-injections?`

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `stagger-temporally?`
- `staggering-factor`

The "staggering factor" is applied to temporal staggering only. It is a constant that multiplies the time step size to give the random sampling interval. The staggering factor controls the percentage of every particle's initial time step that will be sampled. For example, if the staggering factor is 0.5, then the parcels in the injection will be tracked between half and all of their full initial time step. If the staggering factor is 0.1, then the parcels will be tracked between ninety percent and all of their initial time step. If the staggering factor is set to 0.9, the parcels will be tracked between ten percent and all of their initial time step. This allows you to control the amount of smoothing between injections.

The default values for the options in the TUI are spatial staggering for atomizers (including the solid-cone), no temporal staggering and a temporal staggering factor of 1.0. The temporal staggering factor is inactive until the flag for temporal staggering is enabled.

23.3. Setting Initial Conditions for the Discrete Phase

The primary inputs that you must provide for the discrete phase calculations in ANSYS Fluent are the initial conditions that define the starting positions, velocities, and other parameters for each particle stream and the physical effects acting on the particle streams, requiring additional particle properties. You will define the initial conditions for a particle/droplet stream by creating an "injection" and assigning properties to it.

The required initial conditions depend on the injection type, while the physical effects are selected by choosing an appropriate particle type. For some injection types you can provide a particle size distribution, like the Rosin-Rammler distribution, see Using the Rosin-Rammler Diameter Distribution Method (p. 1281).

The initial conditions provide the starting values for all of the dependent discrete phase variables that describe the instantaneous conditions of an individual particle, and include the following:

- position (x, y, z coordinates) of the particle
- velocities (u, v, w) of the particle

  Velocity magnitudes and spray cone angle can also be used (in 3D) to define the initial velocities (see Point Properties for Cone Injections (p. 1271)). For moving reference frames, relative velocities should be specified.

- diameter of the particle, d_p
- temperature of the particle, T_p
- mass flow rate of the particle stream that will follow the trajectory of the individual particle/droplet, $\dot{m}_p$ (required only for coupled calculations)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- additional parameters if one of the atomizer models described in Atomizer Model Theory in the Theory Guide is used for the injection

> **Important**
>
> When an atomizer model is selected, you will not specify initial diameter, velocity, and position quantities for the particles due to the complexities of sheet and ligament breakup. Instead of initial conditions, the quantities you will specify for the atomizer models are global parameters.

These dependent variables (temperature, diameter, and so on) are updated according to the equations of motion (Particle Motion Theory in the Theory Guide) and according to the heat/mass transfer relations applied (Laws for Heat and Mass Exchange in the Theory Guide) as the particle/droplet moves along its trajectory. You can define any number of different sets of initial conditions for discrete phase particles/droplets provided that your computer has sufficient memory.

For the setup of transient particle cases, the point properties for mass flow and velocity can be specified using transient profiles (see Point Properties for Transient Injections (p. 1300)).

For additional information, see the following sections:

23.3.1. Injection Types
23.3.2. Particle Types
23.3.3. Point Properties for Single Injections
23.3.4. Point Properties for Group Injections
23.3.5. Point Properties for Cone Injections
23.3.6. Point Properties for Surface Injections
23.3.7. Point Properties for Plain-Orifice Atomizer Injections
23.3.8. Point Properties for Pressure-Swirl Atomizer Injections
23.3.9. Point Properties for Air-Blast/Air-Assist Atomizer Injections
23.3.10. Point Properties for Flat-Fan Atomizer Injections
23.3.11. Point Properties for Effervescent Atomizer Injections
23.3.12. Point Properties for File Injections
23.3.13. Point Properties for Condensate Injections
23.3.14. Using the Rosin-Rammler Diameter Distribution Method
23.3.15. Creating and Modifying Injections
23.3.16. Defining Injection Properties
23.3.17. Specifying Injection-Specific Physical Models
23.3.18. Specifying Turbulent Dispersion of Particles
23.3.19. Custom Particle Laws
23.3.20. Defining Properties Common to More than One Injection
23.3.21. Point Properties for Transient Injections

23.3.1. Injection Types

ANSYS Fluent provides 11 types of injections:

- single
- group
- cone (only in 3D)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- surface
- plain-orifice atomizer
- pressure-swirl atomizer
- air-blast-atomizer
- flat-fan-atomizer
- effervescent-atomizer
- file
- condensate

For each nonatomizer injection type, you will specify each of the initial conditions listed in Setting Initial Conditions for the Discrete Phase (p. 1265), the type of particle that possesses these initial conditions, and any other relevant parameters for the particle type chosen.

You should create a single injection when you want to specify a single value for each of the initial conditions (Figure 23.9: Particle Injection Defining a Single Particle Stream (p. 1267)). Create a group injection (Figure 23.10: Particle Injection Defining an Initial Spatial Distribution of the Particle Streams (p. 1268)) when you want to define a range for one or more of the initial conditions (for example, a range of diameters or a range of initial positions). To define cone injections in 3D problems, create a cone injection and specify a particle release pattern (point, hollow, ring, or solid) (Figure 23.11: Particle Injection Defining an Initial Spray Distribution of the Particle Velocity (p. 1268)). To release particles from a surface (either a zone surface or a surface you have defined using **Create** in the **Setting Up Domain** ribbon tab (**Surface**group box)), you will create a surface injection. (If you create a surface injection, a particle stream will be released from each facet of the surface. You can use the **Bounded** and **Sample Points** options in the **Plane Surface** dialog box to create injections from a rectangular mesh of particles in 3D (see Plane Surfaces (p. 1722) for details). If you are modeling film condensation, use the **condensate** injection type to define the vapor-liquid material pair and other modeling settings for the film condensation process.

Figure 23.9: Particle Injection Defining a Single Particle Stream

Figure 23.10: Particle Injection Defining an Initial Spatial Distribution of the Particle Streams

Figure 23.11: Particle Injection Defining an Initial Spray Distribution of the Particle Velocity

Particle initial conditions (position, velocity, diameter, temperature, and mass flow rate) can also be read from an external file if none of the injection types listed above can be used to describe your injection distribution.

The inputs for setting injections are described in detail in Defining Injection Properties (p. 1286).

23.3.2. Particle Types

When you define a set of initial conditions (as described in Defining Injection Properties (p. 1286)), you will need to specify the type of particle. The particle types available to you depend on the range of physical models that you have defined.

- A **massless** particle is a discrete element that follows the flow and temperature of the continuous phase. As it has no mass, it has no associated physical properties, and no force is exerted on it. However, you can assign a User-Defined Law to be applied to the massless particle. The massless particle type is available with all ANSYS Fluent models.

- An **inert** particle is a discrete phase element (particle, droplet, or bubble) that obeys the force balance (Equation 16.1 in the Theory Guide) and is subject to heating or cooling via Law 1 (Inert Heating or Cooling (Law 1/Law 6) in the Theory Guide). The inert type is available for all ANSYS Fluent models.

- A **droplet** particle is a liquid droplet in a continuous-phase gas flow that obeys the force balance (Equation 16.1 in the Theory Guide) and that experiences heating/cooling via Law 1 followed by vaporization and boiling via Laws 2 and 3 (Droplet Vaporization (Law 2) and Droplet Boiling (Law 3) in the Theory Guide). The droplet type is available when heat transfer is being modeled and at least two chemical species are active or the non-premixed or partially premixed combustion model is active. You should use the ideal gas law to define the gas-phase density (in the **Create/Edit Materials** dialog box, as discussed in Density Inputs for the Incompressible Ideal Gas Law (p. 484)) when you select the droplet type.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- A **combusting** particle is a solid particle that obeys the force balance (Equation 16.1), and, after an initial phase of inert heating (Law 1), undergoes devolatization (Devolatilization (Law 4)) and then a heterogeneous surface reaction via Law 5 (Surface Combustion (Law 5) in the Theory Guide). Finally, the nonvolatile portion of a combusting particle is subject to inert heating via Law 6. You can also include an evaporating material with the combusting particle by selecting the **Wet Combustion** option in the Set Injection Properties Dialog Box (p. 2796). This allows you to include a material that evaporates and boils via Laws 2 and 3 (Droplet Vaporization (Law 2) and Droplet Boiling (Law 3) in the Theory Guide) before devolatilization of the particle material begins. The combusting type is available when heat transfer is being modeled and at least three chemical species are active or the non-premixed combustion model is active. You should use the ideal gas law to define the gas-phase density (in the Create/Edit Materials Dialog Box (p. 2302)) when you select the combusting particle type.

- A **multicomponent** particle is, as the name implies, a droplet particle containing a mixture of several components or species. The conservation equations of all components, the energy equation, and vapor-liquid-equilibrium at the multicomponent particle surface form a coupled system of differential equations. Law 7, the multicomponent law (Multicomponent Particle Definition (Law 7) in the Theory Guide) is used for such systems. You should use the volume weighted mixing law to define the particle mixture density (in the Create/Edit Materials Dialog Box (p. 2302)) when you select the particle-mixture material type.

Particle Type	Description	Laws Activated
Massless	–	–
Inert	inert/heating or cooling	1, 6
Droplet	heating/evaporation/boiling	1, 2, 3, 6
Combusting	heating; evolution of volatiles/swelling; heterogeneous surface reaction	1, 4, 5, 6
Multicomponent	multicomponent droplets/particles	7

23.3.3. Point Properties for Single Injections

For a single injection, you will define the following initial conditions for the particle stream under the **Point Properties** heading (in the Set Injection Properties Dialog Box (p. 2796)):

- position

  Set the x, y, and z positions of the injected stream along the Cartesian axes of the problem geometry in the **X-**, **Y-**, and **Z-Position** fields. (**Z-Position** will appear only for 3D problems.)

- velocity

  Set the x, y, and z components of the stream's initial velocity in the **X-**, **Y-**, and **Z-Velocity** fields. (**Z-Velocity** will appear only for 3D problems.)

- angular velocity (available when particle rotation is enabled (see Particle Rotation (p. 1291)))

  Set the x, y, and z components of the stream's initial velocity in the **X-**, **Y-**, and **Z-Angular-Velocity** fields. (Only **Z-Angular-Velocity** will appear for 2D problems, because rotation in 2D occurs in the plane, that is, the rotation vector is normal to the plane.)

- diameter

  Set the initial diameter of the injected particle stream in the **Diameter** field.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- temperature

  Set the initial (absolute) temperature of the injected particle stream in the **Temperature** field.

- mass flow rate

  For coupled phase calculations (see Solution Strategies for the Discrete Phase (p. 1325)), set the mass of particles per unit time that follows the trajectory defined by the injection in the **Flow Rate** field. Note that in axisymmetric problems the mass flow rate is defined per 2π radians and in 2D problems per unit meter depth (regardless of the reference value for length).

- duration of injection

  For unsteady particle tracking (see Steady/Transient Treatment of Particles (p. 1242)), set the starting and ending time for the injection in the **Start Time** and **Stop Time** fields. When In-Cylinder mesh motion is enabled, set the starting and ending crank angles for the injection in the **Start Crank Angle** and **Stop Crank Angle** fields. The values you enter are for one complete engine cycle. In subsequent cycles, the injection will be started and stopped according to Equation 10.30 (p. 716), where θ_{event} corresponds to the injection start or end crank angle.

For the massless particle type, you will only need to define the position of the injection. The particle injection velocity is set by the solver equal to the velocity of the continuous phase at the injection point.

If you want to randomly distribute the initial placement of the particles, you can enable **Stagger Positions** and specify **Stagger Radius** to define the particle release region. For more information on spatial staggering, see Staggering of Particles in Space and Time (p. 1264). This option is disabled by default.

23.3.4. Point Properties for Group Injections

For group injections, you will define the properties position, velocity, angular velocity (available when particle rotation is enabled (see Particle Rotation (p. 1291))), diameter, temperature, and flow rate for the **First Point** and **Last Point** in the group. That is, you will define a range of values, ϕ_1 through ϕ_N, for each initial condition ϕ by setting values for ϕ_1 and ϕ_N. ANSYS Fluent assigns a value of ϕ to the i^{th} injection in the group using a linear variation between the first and last values for ϕ:

$$\phi_i = \phi_1 + \frac{\phi_N - \phi_1}{N-1}(i-1) \tag{23.7}$$

Thus, for example, if your group consists of 5 particle streams and you define a range for the initial x location from 0.2 to 0.6 meters, the initial x location of each stream is as follows:

- Stream 1: $x = 0.2$ meters
- Stream 2: $x = 0.3$ meters
- Stream 3: $x = 0.4$ meters
- Stream 4: $x = 0.5$ meters

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Stream 5: $x = 0.6$ meters

Important

In general, you should supply a range for only one of the initial conditions in a given group—leaving all other conditions fixed while a single condition varies among the stream numbers of the group. Otherwise you may find, for example, that your simultaneous inputs of a spatial distribution and a size distribution have placed the small droplets at the beginning of the spatial range and the large droplets at the end of the spatial range.

The specified flow rate is defined per particle stream and can also be interpolated using Equation 23.7 (p. 1270). When a Rosin-Rammler sized distribution is specified the total flow rate will be specified.

For the massless particle type, you will only need to define the first and last point of the injection group position. The particle velocities are set by the solver equal to the velocity of the continuous phase at the injection points.

Note that you can use a different method for defining the size distribution of the particles, as discussed below.

To randomly distribute the initial placement of the particles, enable **Stagger Positions** and specify **Stagger Radius** to define the particle release region. For more information on spatial staggering, see Staggering of Particles in Space and Time (p. 1264). This option is disabled by default.

23.3.5. Point Properties for Cone Injections

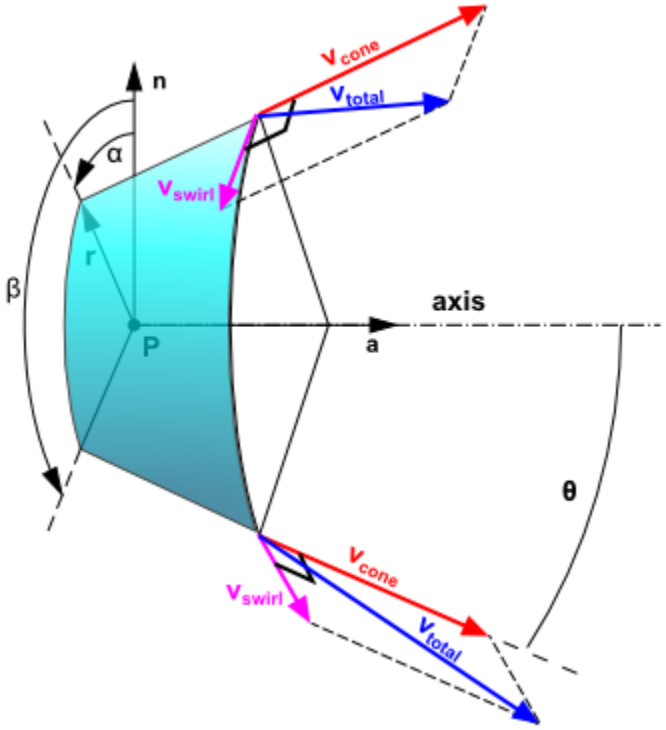
In 3D problems, you can define the following types of particle stream cones:

- **point-cone**
- **hollow-cone**
- **ring-cone**
- **solid-cone**

The particles are released from a 2D region that may be a point, a circle, or an annulus. Its geometry, location, and orientation as well as the shape of the spray cone are defined by the injection parameters. Once you select a cone injection type from the **Cone Type** drop-down list, (in the **Point Properties** tab of the **Set Injection Properties** dialog box), the parameters specific to the selected injection type will be displayed. These inputs are summarized below. Refer to Figure 23.12: Cone Injector Geometry (p. 1272) for an illustration of the geometry.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.12: Cone Injector Geometry



- position

  Set the coordinates of the origin of the spray cone in the **X-**, **Y-**, and **Z-Position** fields (P in Figure 23.12: Cone Injector Geometry (p. 1272))

- diameter

  Set the diameter of the particles in the stream in the **Diameter** field.

- temperature

  Set the temperature of the streams in the **Temperature** field.

- duration of injection

  For unsteady particle tracking (see Steady/Transient Treatment of Particles (p. 1242)), set the starting and ending time for the injection in the **Start Time** and **Stop Time** fields. When In-Cylinder mesh motion is enabled, set the starting and ending crank angles for the injection in the **Start Crank Angle** and **Stop Crank Angle** fields.

- axis

  Set the x, y, and z components of the vector defining the cone's axis in the **X-Axis**, **Y-Axis**, and **Z-Axis** fields. (a in Figure 23.12: Cone Injector Geometry (p. 1272))

- velocity

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Set the velocity magnitude of the particle streams that will be oriented along the specified spray cone in the **Velocity Magnitude** field. (v_{total} in Figure 23.12: Cone Injector Geometry (p. 1272))

Note

This is the total spray velocity magnitude and includes any swirl component in a hollow cone injector.

$$\vec{v}_{total}=\vec{v}_{cone}+\vec{v}_{swirl}$$

- angular velocity

  Set the angular velocity magnitude of the particle streams that will be oriented along the specified spray cone in the **Angular Velocity Magnitude** field.

- cone angle

  Set the included half-angle of the spray cone in the **Cone Angle** field. (θ in Figure 23.12: Cone Injector Geometry (p. 1272))

- radii

  For **hollow-cone**, **ring-cone**, and **solid-cone** injections, specify the **Outer Radius**.

  For **ring-cone** injections, specify a nonzero **Inner Radius** to model injectors that do not emanate from a single point. The particles will be distributed about the axis with the specified radius. (r in Figure 23.12: Cone Injector Geometry (p. 1272))

- swirl fraction (hollow cone only)

  Set the swirl fraction, which determines the relative magnitude of the swirl velocity, in the **Swirl Fraction** field.

  $$SF \equiv \frac{|\vec{v}_{swirl}|}{|\vec{v}_{cone}|+|\vec{v}_{swirl}|}$$

  The direction of the swirl component is defined using the right-hand rule about the cone axis (a negative value for the swirl fraction can be used to reverse the swirl direction).

- mass flow rate

  For coupled calculations, set the total mass flow rate for the streams in the spray cone in the **Total Flow Rate** field. Note that for a 3D sector, the flow rate must be appropriate for the sector defined by the **Azimuthal Start Angle** and **Azimuthal Stop Angle**.

- azimuthal angles

  For 3D sectors, set the **Azimuthal Start Angle** and **Azimuthal Stop Angle** to define the sector. (α and β, respectively, in Figure 23.12: Cone Injector Geometry (p. 1272)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Azimuthal Start Angle** and **Azimuthal Stop Angle** are specified with respect to a reference vector, $\vec{n}$, that is orthogonal to both the cone axis and the global X axis. The reference vector, $\vec{n}$, can be determined as the cross product of the cone axis and the global X axis.

$$\vec{n}=\vec{a}\times\hat{i}$$

where $\hat{i}$ is a unit vector along the global X axis. In the case that the cone axis is along the X axis, then $\vec{n}$ is along the Y axis.

In addition to the above parameters, you can select the following options:

- **Uniform Massflow Distribution** (**solid-cone** or **ring-cone**): Specifies a uniform spacial distribution of mass over the cross-section of a cone. This is especially important when the mesh cell size is smaller than the particle diameter or another characteristic size. This option is enabled by default.
- **Stagger Positions**: Specifies a random spatial distribution of particle steams. For **hollow-cone** injections you must also specify **Stagger Radius** to define the region over which particle release points are staggered in space. For more information on spatial staggering, see Staggering of Particles in Space and Time (p. 1264). This option is disabled by default.

The distribution of the velocity directions in the particle streams for the point-cone, ring-cone, and solid-cone injections is random. Furthermore, duplicating this injection may not necessarily result in the same distribution, at the same location.

Important

For transient calculations, the spatial distribution of streams at the initial injection location is recalculated at each time step. Sampling different possible trajectories allows a more accurate representation of a point, ring, or solid cone using fewer computational parcels. For steady-state calculations, the trajectories are initialized one time and kept the same for subsequent DPM iterations. The trajectories are recalculated when a change in the **Injections** dialog box occurs or when a case and data file are saved. If the residuals and solution change when a small change is made to the injection or when a case and data file are saved, it may mean that there are not enough trajectories being used to represent the point, ring, or solid cone with sufficient accuracy.

Note that you may want to define multiple spray cones emanating from the same initial location in order to specify a known size distribution of the spray or to include a known range of cone angles.

For the massless particle type, you will only need to define position, axis, cone angle, azimuthal angles, and radius. The particle velocities are set by the solver equal to the velocity of the continuous phase at the injection points.

23.3.6. Point Properties for Surface Injections

For surface injections, you will define all the properties described in Point Properties for Single Injections (p. 1269) for single injections except for the initial position of the particle streams. The initial positions of the particles will be the location of the data points on the specified surface(s). Note that you will set the **Total Flow Rate** of all particles released from the surface (required for coupled calculations only). If you want, you can scale the individual mass flow rates of the particles by the ratio of the area of the face they are released from to the total area of the surface. To scale the mass flow rates, select the **Scale Flow Rate By Face Area** option under **Point Properties**. For the massless particle type, you will not

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

need to enter any information to define a surface injection. The particle velocities are set by the solver equal to the velocities of the continuous phase at the injection points.

Note that many surfaces have nonuniform distributions of points. If you want to generate a uniform spatial distribution of particle streams released from a surface in 3D, you can create a bounded plane surface with a uniform distribution using the Plane Surface Dialog Box (p. 2777), as described in Plane Surfaces (p. 1722). In 2D, you can create a rake using the Line/Rake Surface Dialog Box (p. 2727), as described in Line and Rake Surfaces (p. 1718).

In addition to the option of scaling the flow rate by the face area, the normal direction of a face can be used for the injection direction. To use the face normal direction for the injection direction, select the **Inject Using Face Normal Direction** option under **Point Properties** (Figure 23.19: Setting Surface Injection Properties (p. 1288)). Once this option is selected, you only need to specify the velocity magnitude of the injection, not the individual components of the velocity magnitude.

Important

- Only surface injections from boundary surfaces will be moved with the mesh when a sliding mesh or a moving or deforming mesh is being used.
- For moving or deforming mesh simulations, only zonal surfaces can be selected.

A nonuniform size distribution can be used for surface injections, as described below.

23.3.6.1. Using the Rosin-Rammler Diameter Distribution Method

The Rosin-Rammler size distributions described in Using the Rosin-Rammler Diameter Distribution Method (p. 1281) for group injections is also available for surface injections. If you select one of the Rosin-Rammler distributions (**rosin-rammler** or **rosin-rammler-logarithmic**), you will need to specify the following parameters under **Point Properties**, in addition to the initial velocity, temperature, and total flow rate:

- **Min. Diameter**

  This is the smallest diameter to be considered in the size distribution.

- **Max. Diameter**

  This is the largest diameter to be considered in the size distribution.

- **Mean Diameter**

  This is the size parameter, $\overline{d}$, in the Rosin-Rammler equation (Equation 23.9 (p. 1281)).

- **Spread Parameter**

  This is the exponential parameter, n, in Equation 23.9 (p. 1281).

- **Number of Diameters**

  This is the number of diameters in each distribution (that is, the number of different diameters in the stream injected from each face of the surface).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS Fluent will inject streams of particles from each face on the surface, with diameters defined by the Rosin-Rammler distribution function. The total number of injection streams tracked for the surface injection will be equal to the number of diameters in each distribution (**Number of Diameters**) multiplied by the number of faces on the surface.

23.3.7. Point Properties for Plain-Orifice Atomizer Injections

For a plain-orifice atomizer injection, you will define the following initial conditions under **Point Properties**:

- position

  Set the x, y, and z positions of the injected stream along the Cartesian axes of the problem geometry in the **X-Position**, **Y-Position**, and **Z-Position** fields. (**Z-Position** will appear only for 3D problems).

- axis (3D only)

  Set the x, y, and z components of the vector defining the axis of the orifice in the **X-Axis**, **Y-Axis**, and **Z-Axis** fields.

- temperature

  Set the temperature of the streams in the **Temperature** field.

- mass flow rate

  Set the total mass flow rate for the streams in the atomizer in the **Flow Rate** field. Note that in 3D sectors, the flow rate must be appropriate for the sector defined by the **Azimuthal Start Angle** and **Azimuthal Stop Angle**.

- duration of injection

  For unsteady particle tracking (see Steady/Transient Treatment of Particles (p. 1242)), set the starting and ending time for the injection in the **Start Time** and **Stop Time** fields. When In-Cylinder mesh motion is enabled, set the starting and ending crank angles for the injection in the **Start Crank Angle** and **Stop Crank Angle** fields.

- vapor pressure

  Set the vapor pressure governing the flow through the internal orifice (p_v in Table 16.4: List of Governing Parameters for Internal Nozzle Flow in the Theory Guide) in the **Vapor Pressure** field.

- diameter

  Set the diameter of the orifice in the **Injector Inner Diameter** field (d in Table 16.4: List of Governing Parameters for Internal Nozzle Flow in the Theory Guide).

- orifice length

  Set the length of the orifice in the **Orifice Length** field (L in Table 16.4: List of Governing Parameters for Internal Nozzle Flow in the Theory Guide).

- radius of curvature

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Set the radius of curvature of the inlet corner in the **Corner Radius of Curvature** field (r in Table 16.4: List of Governing Parameters for Internal Nozzle Flow in the Theory Guide).

- nozzle parameter

  Set the constant for the spray angle correlation in the **Constant A** field (C_A in Equation 16.331 in the Theory Guide).

- azimuthal angles

  For 3D sectors, set the **Azimuthal Start Angle** and **Azimuthal Stop Angle**.

See The Plain-Orifice Atomizer Model in the Theory Guide for details about how these inputs are used.

If you want to randomly distribute the initial placement of the particles, enable **Stagger Positions** (default). The region over which particle release points will be staggered is defined by the specified orifice diameter. For more information on spatial staggering, see Staggering of Particles in Space and Time (p. 1264).

23.3.8. Point Properties for Pressure-Swirl Atomizer Injections

For a pressure-swirl atomizer injection, you will specify some of the same properties as for a plain-orifice atomizer. In addition to the position, axis (if 3D), temperature, mass flow rate, duration of injection (if unsteady), injector inner diameter, and azimuthal angles (if relevant) described in Point Properties for Plain-Orifice Atomizer Injections (p. 1276), you will need to specify the following parameters under **Point Properties**:

- spray angle

  Set the value of the spray angle of the injected stream in the **Spray Half Angle** field (θ in Equation 16.343 in the Theory Guide).

- pressure

  Set the absolute pressure upstream of the injection in the **Upstream Pressure** field (p_1 in Table 16.4: List of Governing Parameters for Internal Nozzle Flow in the Theory Guide).

- sheet breakup

  Set the value of the empirical constant that determines the length of the ligaments that are formed after sheet breakup in the **Sheet Constant** field ($\ln\left(\frac{\eta_b}{\eta_0}\right)$ in Equation 16.350 in the Theory Guide).

- ligament diameter

  For short waves, set the proportionality constant that linearly relates the ligament diameter, d_L, to the wavelength that breaks up the sheet in the **Ligament Constant** field (see Equation 16.351—Equation 16.354 in the Theory Guide).

- dispersion angle

  For a smooth distribution of the droplets, the initial velocities is varied within this dispersion angle. A sketch of the **Atomizer Dispersion Angle** for a flat fan atomizer is depicted in Figure 23.13: Flat Fan Viewed from Above and from the Side (p. 1280).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See The Pressure-Swirl Atomizer Model in the Theory Guide for details about how these inputs are used.

23.3.9. Point Properties for Air-Blast/Air-Assist Atomizer Injections

For an air-blast/air-assist atomizer, you will specify some of the same properties as for a plain-orifice atomizer. In addition to the position, axis (if 3D), temperature, mass flow rate, duration of injection (if unsteady), injector inner diameter, and azimuthal angles (if relevant) described in Point Properties for Plain-Orifice Atomizer Injections (p. 1276), you will need to specify the following parameters under **Point Properties**:

- outer diameter

  Set the outer diameter of the injector in the **Injector Outer Diameter** field. This value is used in conjunction with the **Injector Inner Diameter** to set the thickness of the liquid sheet (t in Equation 16.337 in the Theory Guide).

- spray angle

  Set the initial trajectory of the film as it leaves the end of the orifice in the **Spray Half Angle** field (θ in Equation 16.343 in the Theory Guide).

- relative velocity

  Set the maximum relative velocity that is produced by the sheet and air in the **Relative Velocity** field.

- sheet breakup

  Set the value of the empirical constant that determines the length of the ligaments that are formed after sheet breakup in the **Sheet Constant** field ($\ln\left(\frac{\eta_b}{\eta_0}\right)$ in Equation 16.350 in the Theory Guide).

- ligament diameter

  For short waves, set the proportionality constant (C_L in Equation 16.353 in the Theory Guide) that linearly relates the ligament diameter, d_L, to the wavelength that breaks up the sheet in the **Ligament Constant** field.

- dispersion angle

  For a smooth distribution of the droplets, the initial velocities is varied within this dispersion angle. A sketch of the **Atomizer Dispersion Angle** for a flat fan atomizer is depicted in Figure 23.13: Flat Fan Viewed from Above and from the Side (p. 1280).

See The Air-Blast/Air-Assist Atomizer Model in the Theory Guide for details about how these inputs are used.

23.3.10. Point Properties for Flat-Fan Atomizer Injections

The flat-fan atomizer model is available only for 3D models. For this type of injection, you will define the following initial conditions under **Point Properties**:

- arc position

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Set the coordinates of the center point of the arc from which the fan originates in the **X-Center**, **Y-Center**, and **Z-Center** fields (see Figure 23.13: Flat Fan Viewed from Above and from the Side (p. 1280)).

- virtual position

  Set the coordinates of the virtual origin of the fan in the **X-Virtual Origin**, **Y-Virtual Origin**, and **Z-Virtual Origin** fields. This point is the intersection of the lines that mark the sides of the fan (see Figure 23.13: Flat Fan Viewed from Above and from the Side (p. 1280)).

- normal vector

  Set the direction that is normal to the fan in the **X-Fan Normal Vector**, **Y-Fan Normal Vector**, and **Z-Fan Normal Vector** fields.

- temperature

  Set the temperature of the streams in the **Temperature** field.

- mass flow rate

  Set the mass flow rate for the streams in the atomizer in the **Flow Rate** field.

- duration of injection

  For unsteady particle tracking (see Steady/Transient Treatment of Particles (p. 1242)), set the starting and ending time for the injection in the **Start Time** and **Stop Time** fields. When In-Cylinder mesh motion is enabled, set the starting and ending crank angles for the injection in the **Start Crank Angle** and **Stop Crank Angle** fields.

- spray half angle

  Set the initial half angle of the drops as they leave the end of the orifice in the **Spray Half Angle** field.

- orifice width

  Set the width of the orifice (in the normal direction) in the **Orifice Width** field.

- sheet breakup

  Set the value of the empirical constant that determines the length of the ligaments that are formed after sheet breakup in the **Flat Fan Sheet Constant** field (see Equation 16.350 in the Theory Guide).

- dispersion angle

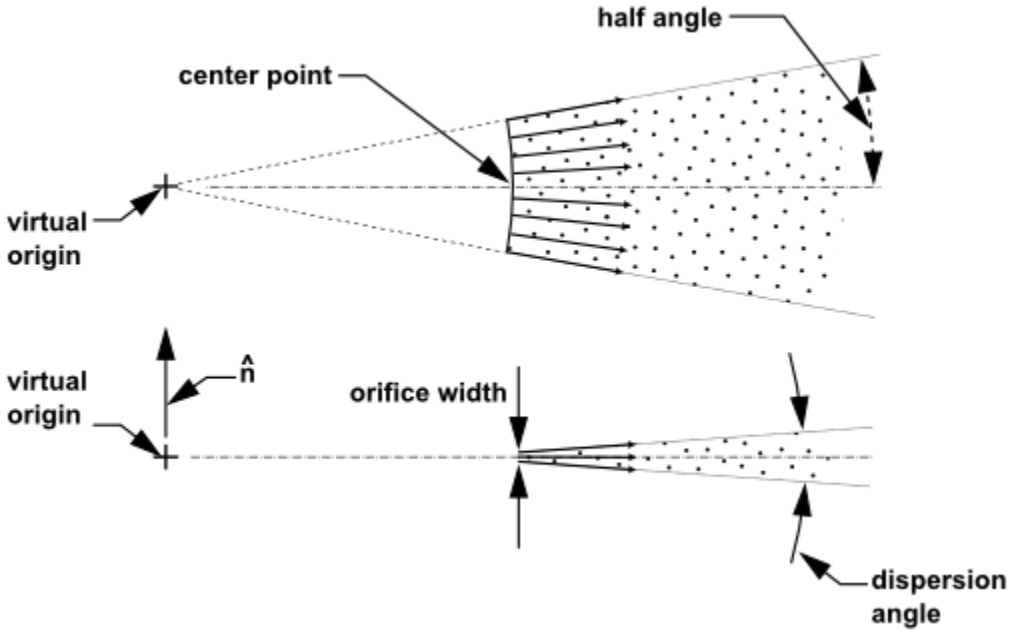
  For a smooth distribution of the droplets, the initial velocities is varied within this dispersion angle. A sketch of the **Atomizer Dispersion Angle** is depicted in Figure 23.13: Flat Fan Viewed from Above and from the Side (p. 1280).

See The Flat-Fan Atomizer Model in the Theory Guide for details about how these inputs are used.

To randomly distribute the initial placement of the particles, enable **Stagger Positions** (default). The orifice width is used as radius to define the region over which particle release points will be staggered in space. For more information on spatial staggering, see Staggering of Particles in Space and Time (p. 1264).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.13: Flat Fan Viewed from Above and from the Side



23.3.11. Point Properties for Effervescent Atomizer Injections

For an effervescent atomizer injection, you will specify some of the same properties as for a plain-orifice atomizer. In addition to the position, axis (if 3D), temperature, mass flow rate (including both flashing and non-flashing components), duration of injection (if unsteady), vapor pressure, injector inner diameter, and azimuthal angles (if relevant) described in Point Properties for Plain-Orifice Atomizer Injections (p. 1276), you will need to specify the following parameters under **Point Properties**:

- mixture quality

  Set the mass fraction of the injected mixture that vaporizes in the **Mixture Quality** field (x in Equation 16.360 in the Theory Guide).

- saturation temperature

  Set the saturation temperature of the volatile substance in the **Saturation Temp.** field.

- droplet dispersion

  Set the parameter that controls the spatial dispersion of the droplet sizes in the **Dispersion Constant** field (C_{eff} in Equation 16.360 in the Theory Guide).

- spray angle

  Set the initial trajectory of the film as it leaves the end of the orifice in the **Maximum Half Angle** field.

See The Effervescent Atomizer Model in the Theory Guide for details about how these inputs are used.

23.3.12. Point Properties for File Injections

The file for a file injection has the following form:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
(( x y z u v w diameter temperature mass-flow) name )
```

with all of the parameters in SI units. All the parentheses are required, but the name is optional.

Sample files generated during sampling of trajectories for steady particles (see Sampling of Trajectories (p. 1355)) can also be used as injection files since they have a similar file format.

23.3.13. Point Properties for Condensate Injections

For a condensate injection, you will define only the duration of the injection by setting the starting and ending time for the injection in the **Start Time** and **Stop Time** fields under the **Point Properties** tab in the **Set Injection Properties** dialog box. The duration of the injection determines the time during which film condensation will be initiated on the walls that have the **Film Condensation** option enabled.

23.3.14. Using the Rosin-Rammler Diameter Distribution Method

For liquid sprays, a convenient representation of the droplet size distribution is the Rosin-Rammler expression. The complete range of sizes is divided into an adequate number of discrete intervals; each represented by a mean diameter for which trajectory calculations are performed. If the size distribution is of the Rosin-Rammler type, the mass fraction of droplets of diameter greater than d is given by

$$Y_d = e^{-(d/\bar{d})^n} \tag{23.8}$$

where $\bar{d}$ is the size constant and n is the size distribution parameter.

By default, you will define the size distribution of particles by inputting a diameter for the first and last points and using the linear equation (Equation 23.7 (p. 1270)) to vary the diameter of each particle stream in the group. When you want a different mass flow rate for each particle/droplet size, however, the linear variation may not yield the distribution you need. Your particle size distribution may be defined most easily by fitting the size distribution data to the Rosin-Rammler equation. In this approach, the complete range of particle sizes is divided into a set of discrete size ranges, each to be defined by a single stream that is part of the group. Assume, for example, that the particle size data obeys the following distribution:

Diameter Range (μm)	Mass Fraction in Range
0–70	0.05
70–100	0.10
100–120	0.35
120–150	0.30
150–180	0.15
180–200	0.05

The Rosin-Rammler distribution function is based on the assumption that an exponential relationship exists between the droplet diameter, d, and the mass fraction of droplets with diameter greater than d, Y_d:

$$Y_d = e^{-(d/\bar{d})^n} \tag{23.9}$$

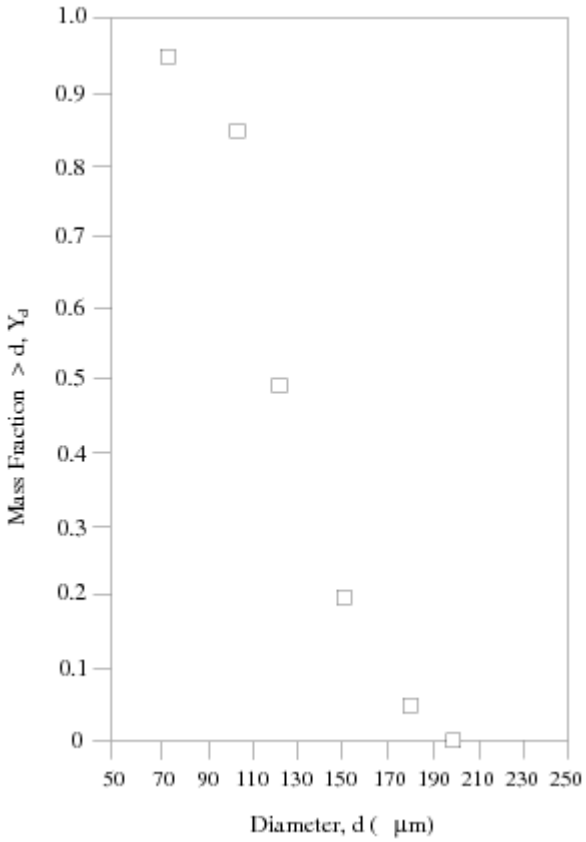
ANSYS Fluent refers to the quantity $\bar{d}$ in Equation 23.9 (p. 1281) as the **Mean Diameter** and to n as the **Spread Parameter**. These parameters are specified by you (in the Set Injection Properties Dialog Box (p. 2796) under the **First Point** heading) to define the Rosin-Rammler size distribution. To solve for

these parameters, you must fit your particle size data to the Rosin-Rammler exponential equation. To determine these inputs, first recast the given droplet size data in terms of the Rosin-Rammler format. For the example data provided above, this yields the following pairs of d and Y_d:

Diameter, d (μm)	**Mass Fraction with Diameter Greater than d, Y_d**
70	0.95
100	0.85
120	0.50
150	0.20
180	0.05
200	(0.00)

A plot of Y_d vs. d is shown in Figure 23.14: Example of Cumulative Size Distribution of Particles (p. 1282).

Figure 23.14: Example of Cumulative Size Distribution of Particles



Next, derive values of $\bar{d}$ and n such that the data in Figure 23.14: Example of Cumulative Size Distribution of Particles (p. 1282) fit Equation 23.9 (p. 1281). The value for $\bar{d}$ is obtained by noting that this is the value of d at which $Y_d = e^{-1} \approx 0.368$. From Figure 23.14: Example of Cumulative Size Distribution of Particles (p. 1282), you can estimate that this occurs for $d \approx 131 \mu$m. The numerical value for n is given by

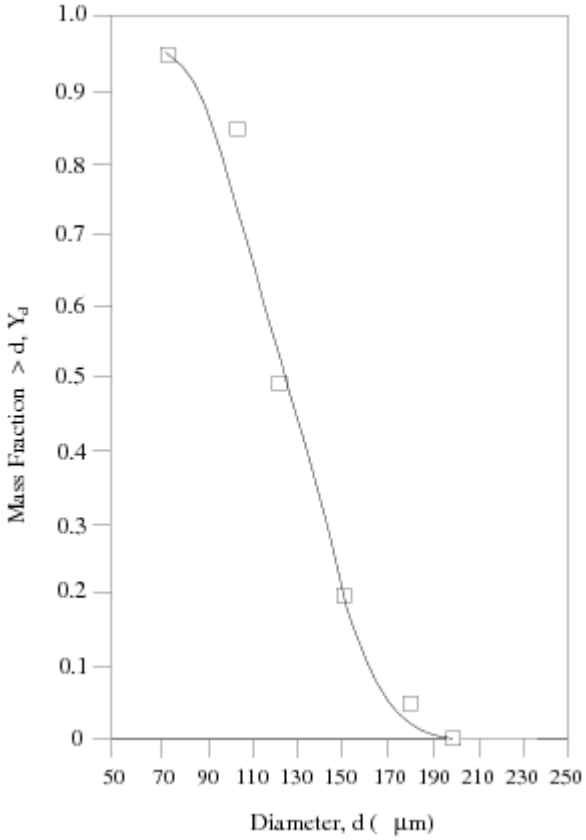
$$n = \frac{\ln\left(-\ln Y_d\right)}{\ln\left(d/\bar{d}\right)} \tag{23.10}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

By substituting the given data pairs for Y_d and $d / \overline{d}$ into this equation, you can obtain values for n and find an average. Doing so yields an average value of $n = 4.52$ for the example data above. The resulting Rosin-Rammler curve fit is compared to the example data in Figure 23.15: Rosin-Rammler Curve Fit for the Example Particle Size Data (p. 1283). You can enter values for $\overline{d}$ and n, as well as the diameter range of the data and the total mass flow rate for the combined individual size ranges, using the Set Injection Properties Dialog Box (p. 2796).

This technique of fitting the Rosin-Rammler curve to spray data is used when reporting the Rosin-Rammler diameter and spread parameter in the **Discrete Phase Summary** dialog box in Summary Reporting of Current Particles (p. 1358).

Figure 23.15: Rosin-Rammler Curve Fit for the Example Particle Size Data



A second Rosin-Rammler distribution is also available based on the natural logarithm of the particle diameter. If in your case, the smaller-diameter particles in a Rosin-Rammler distribution have higher mass flows in comparison with the larger-diameter particles, you may want better resolution of the smaller-diameter particle streams, or "bins". You can therefore choose to have the diameter increments in the Rosin-Rammler distribution done uniformly by $\ln d$.

In the standard Rosin-Rammler distribution, a particle injection may have a diameter range of 1 to 200 μm. In the logarithmic Rosin-Rammler distribution, the same diameter range would be converted to a range of $\ln 1$ to $\ln 200$, or about 0 to 5.3. In this way, the mass flow in one bin would be less-heavily skewed as compared to the other bins.

When a Rosin-Rammler size distribution is being defined for the group of streams, you should define (in addition to the initial velocity, position, and temperature) the following parameters, which appear under the heading for the **First Point**:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Total Flow Rate**

  This is the total mass flow rate of the N streams in the group. Note that in axisymmetric problems this mass flow rate is defined per 2π radians and in 2D problems per unit meter depth.

- **Min. Diameter**

  This is the smallest diameter to be considered in the size distribution.

- **Max. Diameter**

  This is the largest diameter to be considered in the size distribution.

- **Mean Diameter**

  This is the size parameter, $\overline{d}$, in the Rosin-Rammler equation (Equation 23.9 (p. 1281)).

- **Spread Parameter**

  This is the exponential parameter, n, in Equation 23.9 (p. 1281).

23.3.14.1. The Stochastic Rosin-Rammler Diameter Distribution Method

For atomizer injections, a Rosin-Rammler distribution is assumed for the particles exiting the injector. In order to decrease the number of particles necessary to accurately describe the distribution, the diameter distribution function is randomly sampled for each instance where new particles are introduced into the domain.

The Rosin-Rammler distribution can be written as

$$1-Y=\exp\left[-\left(\frac{D}{\overline{d}}\right)^{n}\right] \tag{23.11}$$

where Y is the mass fraction smaller than a given diameter D, $\overline{d}$ is the Rosin-Rammler diameter and n is the Rosin-Rammler exponent. This expression can be inverted by taking logs of both sides and rearranging,

$$D=\overline{d}\left(-\ln\left(1-Y\right)\right)^{1/n}. \tag{23.12}$$

Given a mass fraction Y along with parameters $\overline{d}$ and n, this function will explicitly provide a diameter, D. Diameters for the atomizer injectors described in Point Properties for Plain-Orifice Atomizer Injections (p. 1276) are obtained by uniformly sampling Y in Equation 23.12 (p. 1284).

23.3.15. Creating and Modifying Injections

You will use the **Injections** branch of the tree and the Injections Dialog Box (p. 2716) (Figure 23.17: The Injections Dialog Box (p. 1285), Figure 23.16: The Injections Branch of the Tree (p. 1285)) to create, copy, delete, list, read, and write injections.

Setup → Models → Discrete Phase → Injections → New...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.16: The Injections Branch of the Tree

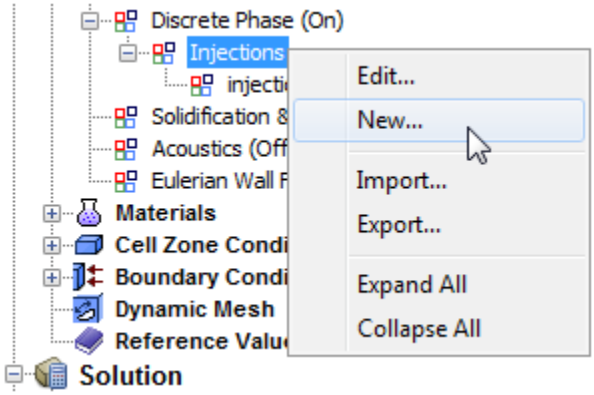
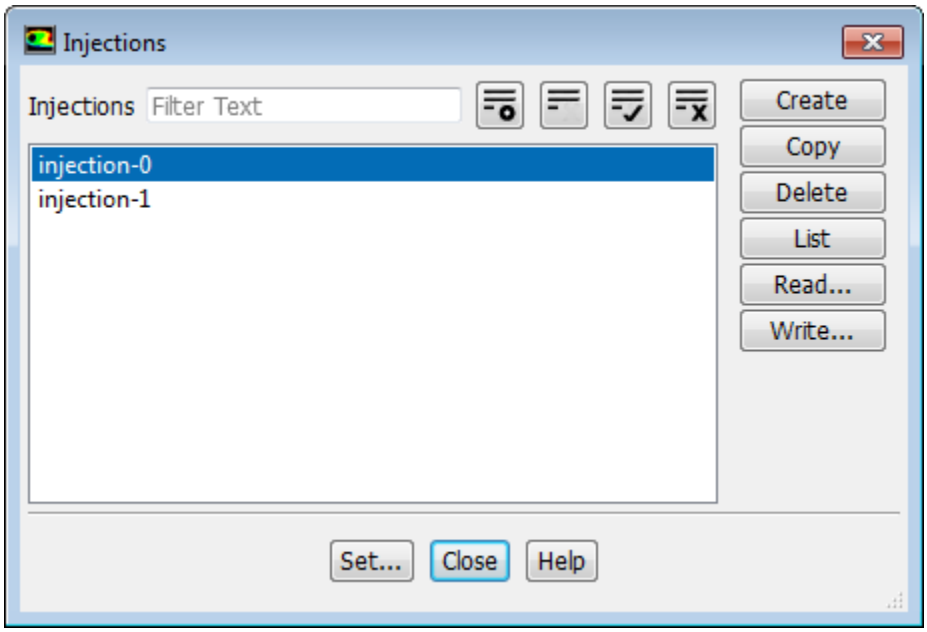


Figure 23.17: The Injections Dialog Box



(You can also click the **Injections...** button in the Discrete Phase Model Dialog Box (p. 2278) to open the **Injections** dialog box.)

23.3.15.1. Creating Injections

To create an injection, click the **Create** button. The Set Injection Properties Dialog Box (p. 2796) will open automatically to allow you to set the injection properties (as described in Defining Injection Properties (p. 1286)). After the injection is created, the new injection will appear in the **Injections** list, in the **Injections** dialog box.

23.3.15.2. Modifying Injections

To modify an existing injection, select its name in the **Injections** list and click the **Set...** button. The **Set Injection Properties** dialog box will open, and you can modify the properties as needed.

If you have two or more injections for which you want to set some of the same properties, select their names in the **Injections** list and click the **Set...** button. The **Set Multiple Injection Properties** dialog

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

box will open, which will allow you to set the common properties. For instructions about using this dialog box, see Defining Properties Common to More than One Injection (p. 1297).

23.3.15.3. Copying Injections

To copy an existing injection to a new injection, select the existing injection in the **Injections** list and click the **Copy** button. The Set Injection Properties Dialog Box (p. 2796) will open with a new injection that has the same properties as the injection you selected. This is useful if you want to set another injection with similar properties.

23.3.15.4. Deleting Injections

You can delete an injection by selecting its name in the **Injections** list and clicking the **Delete** button.

23.3.15.5. Listing Injections

To list the initial conditions for the particle streams in the selected injection, click the **List** button. ANSYS Fluent reports the initial conditions (in SI units) in the console under various columns:

- The particle stream number is in the column headed `NO`.
- The particle type (`IN` for inert, `DR` for droplet, or `CP` for combusting particle) is in the column headed `TYP`.
- The x, y, and z positions are in the columns headed `(X)`, `(Y)`, and `(Z)`.
- The x, y, and z velocities are in the columns headed `(U)`, `(V)`, and `(W)`.
- The temperature is in the column headed `(T)`.
- The diameter is in the column headed `(DIAM)`.
- The mass flow rate in the column headed `(MFLOW)`.

23.3.15.6. Reading and Writing Injections

To transfer information about DPM injections from one case file to another, use the **Read...** and **Write...** buttons. You can write selected injections to a file, which can be read into a different ANSYS Fluent session, simplifying the setup of new case files. To write the injection, select the injection from the list, then click **Write...**. The Select File Dialog Box (p. 19) will open where you can enter the name of your injection file. To read in an injection, click **Read...** to open The Select File Dialog Box (p. 19) where you will select the injection file to read in. If the injection that you imported has the same name as that in your current case, then ANSYS Fluent will rename the imported injection.

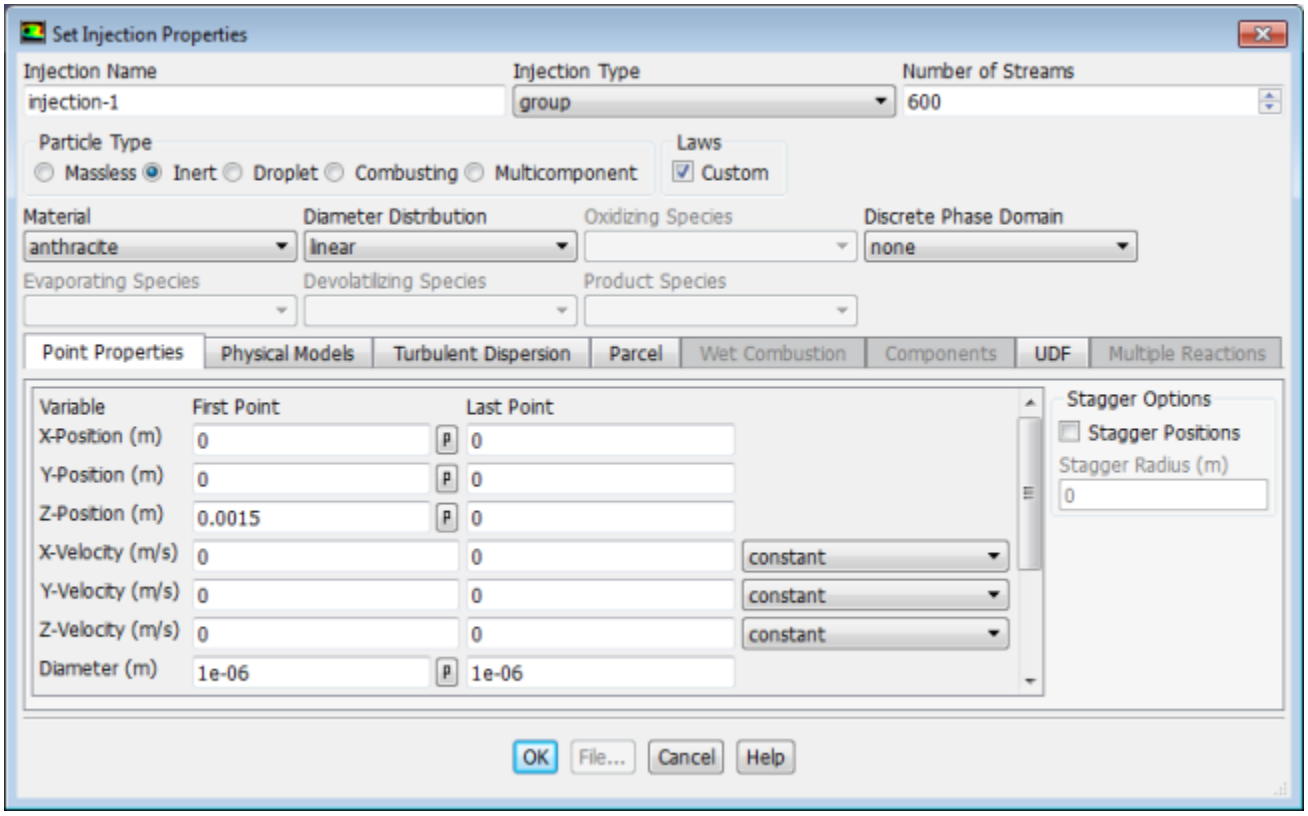
After reading injections, you may need to visit the **Injections** dialog box to modify the settings for the injection material and the DPM laws since the presumed settings may have changed in the current case file setup.

23.3.16. Defining Injection Properties

Once you have created an injection (using the Injections Dialog Box (p. 2716), as described in Creating and Modifying Injections (p. 1284)), you will use the Set Injection Properties Dialog Box (p. 2796) (Figure 23.18: The Set Injection Properties Dialog Box (p. 1287)) to define the injection properties. (Remember that this dialog box will open when you create a new injection, or when you select an existing injection and click the **Set...** button in the **Injections** dialog box.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.18: The Set Injection Properties Dialog Box



The procedure for defining an injection is as follows:

1. If you want to change the default name of the injection, enter a new name in the **Injection Name** field. This is recommended if you are defining a large number of injections so you can easily distinguish them. When assigning names to your injections, keep in mind the selection shortcut described in Creating and Modifying Injections (p. 1284).

2. Choose the type of injection in the **Injection Type** drop-down list. The injection choices (**single**, **group**, **cone**, **surface**, **plain-orifice-atomizer**, **pressure-swirl-atomizer**, **air-blast-atomizer**, **flat-fan-atomizer**, **effervescent-atomizer**, **condensate**, and **file**) are described in Injection Types (p. 1266). Note that if you select any of the atomizer models, you also must set the **Viscosity** and **Droplet Surface Tension** in the **Create/Edit Materials** dialog box.

   Important

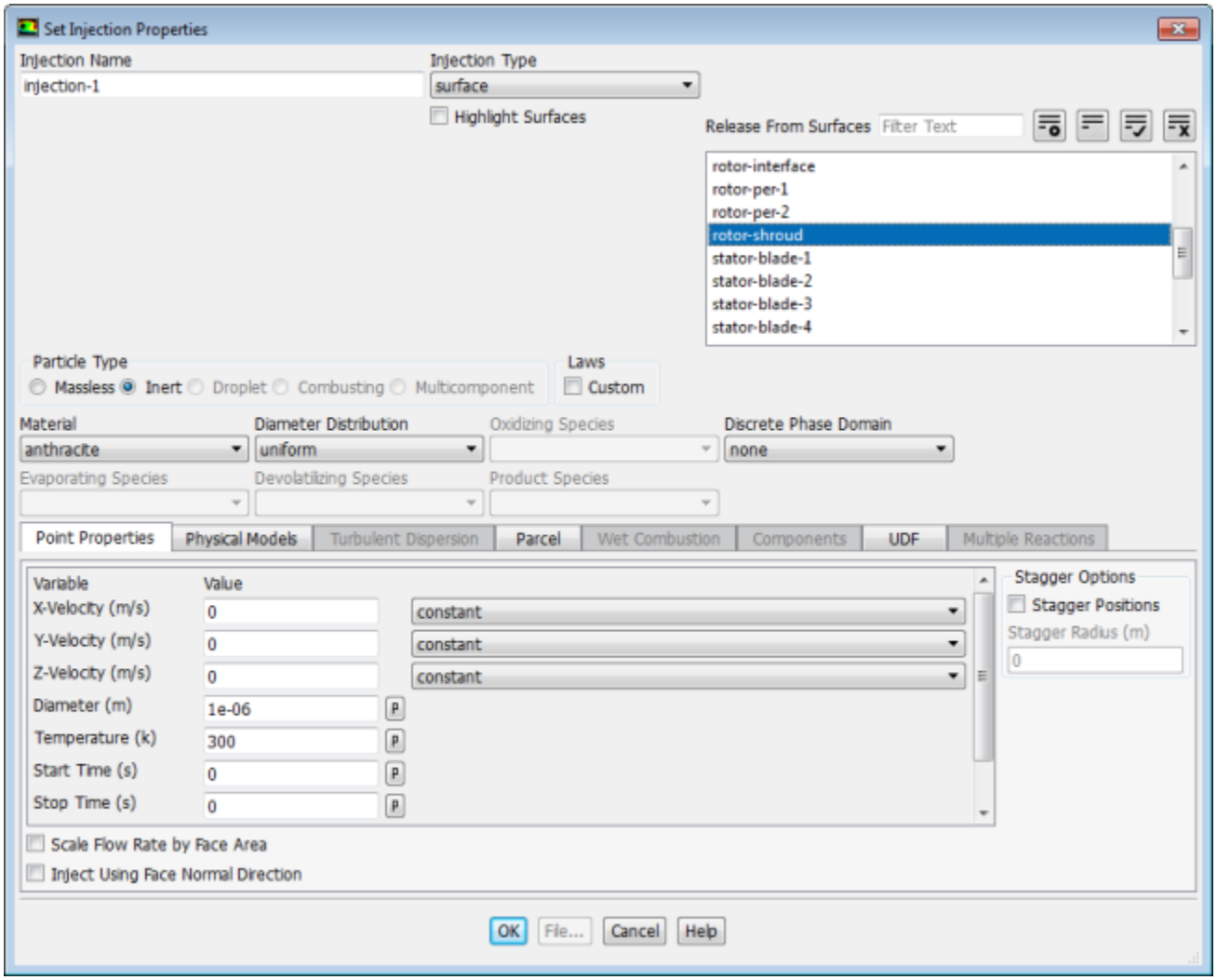
   Note that only surface injections from boundary surfaces will be moved with the mesh when a sliding mesh or a moving or deforming mesh is being used.

3. If you are defining a **single** or **condensate** injection, go to the next step. For a **group**, **cone**, or any of the **atomizer** injections, set the **Number of Streams** in the group, spray cone, or atomizer.

   If you are defining a **surface** injection (see Figure 23.19: Setting Surface Injection Properties (p. 1288)), choose the surface(s) from which the particles will be released in the **Release From Surfaces** list. If you are reading the injection from a file, click the **File...** button at the bottom of the Set Injection Properties Dialog Box (p. 2796) and specify the file to be read in the resulting **Select File** dialog box. The parameters in the injection file must be in SI units.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.19: Setting Surface Injection Properties



4. Select **Massless**, **Inert**, **Droplet**, **Combusting**, or **Multicomponent** as the **Particle Type**. The available types are described in Particle Types (p. 1268). Note that for the **condensate** injection, the only available particle types are **Droplet** and **Multicomponent**.

5. Choose the material for the particle(s) in the **Material** drop-down list.

   If this is the first time you have created a particle of this type, you can choose from all of the materials of this type defined in the database. If you have already created a particle of this type, the only available material will be the material you selected for that particle. You can define additional materials by copying them from the database or creating them from scratch, as discussed in Setting Discrete-Phase Physical Properties (p. 1316) and described in detail in Using the **Create/Edit Materials** Dialog Box (p. 461).

   Important

   Note that you will not choose a **Material** for a **Massless** particle type.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. If you are defining a **group**, **cone**, or **surface** injection and you want to change from the default **linear** (for group injections) or **uniform** (for cone and surface injections) interpolation method used to determine the size of the particles, select **rosin-rammler** or **rosin-rammler-logarithmic** in the **Diameter Distribution** drop-down list. The Rosin-Rammler method for determining the range of diameters for a group injection is described in Using the Rosin-Rammler Diameter Distribution Method (p. 1281).

7. If you have created a customized particle law using user-defined functions, enable the **Custom** option under **Laws** and specify the appropriate laws as described in Custom Particle Laws (p. 1296).

8. If your particle type is **Inert**, go to the next step. If you are defining **Droplet** particles, select the gas phase species created by the vaporization and boiling laws (Laws 2 and 3) in the **Evaporating Species** drop-down list.

   If you are defining **Combusting** particles, select the gas phase species created by the devolatilization law (Law 4) in the **Devolatilizing Species** drop-down list, the gas phase species that participates in the surface char combustion reaction (Law 5) in the **Oxidizing Species** list, and the gas phase species created by the surface char combustion reaction (Law 5) in the **Product Species** list. Note that if the **Combustion Model** for the selected combusting particle material (in the **Create/Edit Materials** dialog box) is the **multiple-surface-reaction** model, then the **Oxidizing Species** and **Product Species** lists will be disabled because the reaction stoichiometry has been defined in the mixture material.

   If you are defining **Multicomponent** particles, maw 7 will go into effect. Notice that the **Components** tab will become active when this particle type is selected. See below for information on the **Components** tab.

9. Click the **Point Properties** tab (the default), and specify the point properties (position, velocity, diameter, temperature, and—if appropriate—mass flow rate and any atomizer-related parameters) as described for each injection type in Point Properties for Single Injections (p. 1269) – Point Properties for Effervescent Atomizer Injections (p. 1280).

   For surface injections, you can enable the **Scale Flow Rate by Face Area** and you can choose the injection direction. To use the face normal direction for the injection direction, select the **Inject Using Face Normal Direction** option under **Point Properties** (Figure 23.19: Setting Surface Injection Properties (p. 1288)). Once this option is selected, you only need to specify the velocity magnitude of the injection, not the individual components of the velocity magnitude.

10. If you want to set up injection-specific physics models such as drag laws, Brownian motion, and breakup, click the **Physical Models** tab and configure the injection physics models as described in Specifying Injection-Specific Physical Models (p. 1290).

11. If the flow is turbulent and you want to include the effects of turbulence on the particle dispersion, click the **Turbulent Dispersion** tab, enable the **Discrete Random Walk Model** under **Stochastic tracking** or the **Cloud Model**, and set the related parameters as described in Specifying Turbulent Dispersion of Particles (p. 1294).

12. If you have enabled **Unsteady Particle Tracking**, you can define settings for how the parcels are released. Click the **Parcel** tab and select a **Parcel Release Method** from the drop-down list. The default method is **standard**, in which a single parcel is released per injection stream per time step. Alternatively can choose **constant-number**, **constant-mass**, or **constant-diameter**. Refer to Steady/Transient Treatment of Particles (p. 1242) for details of these methods.

13. If your combusting particle includes an evaporating material, click the **Wet Combustion** tab, select the **Wet Combustion Model** option, and then select the material that is evaporating/boiling from the particle

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

before devolatilization begins in the **Liquid Material** drop-down list. You should also set the volume fraction of the liquid present in the particle by entering the value of the **Liquid Fraction**. Finally, select the gas phase species created by the evaporating and boiling laws in the **Evaporating Species** drop-down list in the top part of the dialog box.

14. If you include multicomponent droplets as the material in your discrete phase model, the **Components** tab will become active. In this tab, you will specify the **Mass Fraction** of each of the components. Note that the sum of the mass fractions should add up to unity, otherwise ANSYS Fluent will adjust the values such that you have a sum of 1 for the mass fraction, and will prompt you to accept the entry. Under **Evaporating Species**, select **not-vaporizing** if the component in the particle does not vaporize. Otherwise, select the species that will be vaporized.

    To change the components of the multicomponent droplet:

    a. In the **Create/Edit Materials** dialog box, either define a new droplet materials or copy appropriate materials from the Fluent materials database.

    b. Click **Edit...** next to **Mixture Species**.

    c. In the **Species** dialog box, add the droplet materials that you have copied or defined to the **Selected Species** list and remove unwanted materials.

15. If you want to use a user-defined function to initialize the injection properties, click the **UDF** tab to access the UDF inputs. You can select an **Initialization** function under **User-Defined Functions** to modify injection properties at the time the particles are injected into the domain. This allows the position and/or properties of the injection to be set as a function of flow conditions. More information about user-defined functions can be found in the Fluent Customization Manual.

16. If you have defined more than one particle surface species, for example, carbon (**C<s>**) and sulfur (**S<s>**), you will need to specify the mass fraction of each particle surface species in the combusting particle. To do so, click the **Multiple Reactions** tab, and enter the **Species Mass Fractions**. These mass fractions refer to the combustible fraction of the combusting particle, and should sum to 1. If there is only one surface species in the mixture material, the mass fraction of that species will be set to 1, and you will not specify anything under **Multiple Surface Reactions**.

23.3.17. Specifying Injection-Specific Physical Models

Drag and breakup models can be specified on a per-injection basis, allowing you to specify the most appropriate models for each injection in your model. These per-injection models are specified on the **Physical Models** tab of the **Set Injection Properties** dialog box.

23.3.17.1. Drag Laws

The **spherical**, **nonspherical**, **Stokes-Cunningham**, and **high-Mach-number** laws described in Particle Force Balance in the Theory Guide are always available, and the **dynamic-drag** law described in Dynamic Drag Model Theory in the Theory Guide is available only when one of the droplet breakup models is used in conjunction with unsteady tracking. See Breakup (p. 1292) for information about enabling the droplet breakup models. The **Grace** and **Ishii-Zuber** drag laws described in Bubbly Flow Drag Laws in the *Fluent Theory Guide* allow to model the drag coefficient in gas-liquid flows in which the bubbles can have a range of different shapes. Both drag laws are only available if the magnitude and direction of the gravity vector are specified in the **Operating Conditions** dialog box. The remaining three, **Wen-Yu**, **Gidaspow**, and **Syamlal-OBrien** are available only when the dense discrete phase model is enabled (Including the Dense Discrete Phase Model (p. 1481)) and the flow regime consists of a dense gas-solid.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

However, with these models, you cannot verify whether it really is a dense flow or a gas-solid flow. It is up to you to decide. In any case, these drag formulations are suitable for dense gas-solid flows.

If the **spherical**, **high-Mach-number**, **dynamic-drag**, **Wen-Yu**, **Gidaspow**, or **Syamlal-OBrien** drag law is selected, no further inputs are required. If the **nonspherical** law is selected, the particle **Shape Factor** (ϕ in Equation 16.75 in the Theory Guide) must be specified. The shape factor value cannot exceed 1. For the **Stokes-Cunningham** law, the **Cunningham Correction** factor (C_c in Equation 16.77 in the Theory Guide) must be specified.

23.3.17.2. Particle Rotation

To set-up particle rotation, follow the steps below:

1. In the **Set Injection Properties** dialog box, under the **Physical Models** tab, enable **Particle Rotation**.

   The options related to the particle rotation feature appear in the **Set Injection Properties** dialog box.

2. From the **Rotational Drag Law** drop-down list, select the model for calculating the rotational drag coefficient C_ω. The following options are available:

   Dennis-et-al
   Uses Equation 16.4 in the *Fluent Theory Guide* to calculate the rotational drag coefficient C_ω.

   none (default)
   Implies zero torque, that is, it excludes the influence of the fluid on the rotational particle motion.

3. From the **Magnus Lift Law** drop-down list, select the model for calculating the rotational lift coefficient C_{RL}. The following models are available:

   Oesterle-Bui-Dinh
   Uses Oesterle and Bui Dinh's formulation (Equation 16.18 in the *Fluent Theory Guide*).

   Tsuji-et-al
   Is based on the particle spin rate (Equation 16.19 in the *Fluent Theory Guide*).

   Rubinow-Keller
   Uses a linear dependence of the rotational lift coefficient C_{RL} on the spin rate (Equation 16.21 in the *Fluent Theory Guide*).

   none (default)
   Implies that the Magnus lift will not be included in your simulation.

4. Under the **Point Properties** tab, specify angular velocity components for **single**, **group**, and **surface** injections; and angular velocity magnitudes for **cone** injections.

5. For each wall with a **reflect** boundary condition for the discrete phase, specify the friction coefficient (μ in Equation 16.209 and Equation 16.214 in the *Fluent Theory Guide*) under the **DPM** tab of the **Wall** dialog box. For details, see Friction Coefficient (p. 1305).

The particle rotation model options are also accessible through the text user interface, under `define/models/dpm/injections/injection-properties/set/physical-models/particle-rotation`. Note that the `physical-models` command becomes available only

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

after you have selected the injection(s) using the prompts of the `define/models/dpm/injections/properties/set/pick-injections-to-set` text command.

23.3.17.3. Rough Wall Model

To use the rough wall model for walls with the **reflect** boundary condition type:

1. In the **Injection Settings** dialog box, under the **Physical Models** tab, select **Rough Wall Model**.
2. In the **Wall** boundary conditions dialog box for the appropriate wall, under the **DPM** tab, select **reflect** from the **Boundary Cond. Type** drop-down list.
3. Specify the following wall roughness parameters:

   Ra
   is the wall mean roughness (R_a in Equation 16.216 in the *Fluent Theory Guide*)

   Rq
   is the standard deviation of the roughness structure (R_q in Equation 16.216 in the *Fluent Theory Guide*)

   RSm
   is the mean roughness slope (the average distance between peaks of wall material) (RS_m in Equation 16.216 in the *Fluent Theory Guide*)

   Note

   Note that the **DPM Wall Roughness Parameters** will only appear in the **Wall** dialog box with the `reflect` DPM boundary condition type if the **Rough Wall Model** is selected in the **Injection Settings** dialog box.

For information about the rough wall model, see Rough Wall Model in the *Fluent Theory Guide*.

23.3.17.4. Brownian Motion Effects

For sub-micron particles in laminar flow, you may want to include the effects of Brownian motion (described in Brownian Force in the Theory Guide) on the particle trajectories. To do so, enable the **Brownian Motion** option under the **Physical Models** tab. In order to include Brownian motion effects, you must also select the **Stokes-Cunningham** drag law in the **Drag Law** drop-down list under **Drag Parameters**, and specify the **Cunningham Correction** (C_c in Equation 16.77 in the Theory Guide).

23.3.17.5. Breakup

To enable the modeling of particle breakup for the injection, select **Enable Breakup** and choose the **Breakup Model** (**TAB**, **Wave**, **KHRT**, or **SSD**) from the drop-down list. A detailed description of these models can be found in Secondary Breakup Model Theory in the Theory Guide.

- For the **TAB** model, you must specify the following values:
  - **y0** is the initial distortion at time equal to zero (y_0 in Equation 16.367 in the Theory Guide). The default value ($y_0=0$) is recommended.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  – The number of **Breakup Parcels** (under the **Breakup Model** drop-down list), to split the droplet into several child parcels, as described in Velocity of Child Droplets in the Theory Guide. The diameter of the child parcels is sampled from a Rosin-Rammler distribution. This can be switched off in the TUI with the command:

    ```
    /define/models/dpm/spray-model/tab-randomize-breakup-parcel-diameter?
    ```

- For the **Wave** model, you must specify the following values:

  – **B0** is the constant B_0 in Equation 16.397 in the Theory Guide.

  – **B1** is the constant B_1 in Equation 16.399 in the Theory Guide.

  Note

  You will generally not need to modify the value of **B0**, as the default value 0.61 is acceptable for nearly all cases. A value of 1.73 is recommended for **B1**.

  The **Wave** model implementation has been formulated to deal with the initial breakup of a cylindrical liquid jet. In the Rayleigh regime (that is, at low gas Weber numbers), a cylindrical jet breaks up into droplets whose diameters are larger than that of the jet itself. In this regime, the model assumes that the diameter of the continuous, cylindrical liquid jet has been entered as initial diameter in the injection. Therefore, the droplet diameter can grow beyond the initial diameter. For details, see Droplet Breakup in the *Fluent Theory Guide*.

  If you want to suppress this feature of the Wave model, use the TUI command:

  ```
  /define/models/dpm/spray-model/wave-allow-rayleigh-growth? no
  ```

  Note that this takes effect unconditionally for all injections using the **Wave** or **KHRT** breakup models.

- For the **KHRT** model, you must specify the following values:

  – **B0** is the constant B_0 in Equation 16.397 in the Theory Guide.

  – **B1** is the constant B_1 in Equation 16.399 in the Theory Guide.

  – **Ctau** is the constant C_τ in Equation 16.404 in the Theory Guide.

  – **CRT** is the constant C_{RT} in Equation 16.405 in the Theory Guide.

  – **CL** is the constant C_L in Equation 16.400 in the Theory Guide.

  The constants **B0** and **B1** are the same as for the Wave model.

  As for the **Wave** model described above, the **KHRT** model is formulated to deal with the initial breakup of a cylindrical liquid jet. The behavior in the Rayleigh regime is as described above for the **Wave** model and the same TUI command can be used to modify the behavior. In addition, the **KHRT** model uses the Liquid Core Approximation as described in Liquid Core Length in the *Fluent Theory Guide*. To suppress the use of the Liquid Core Approximation you can set the Levich constant, C_L, to zero.

- For the **SSD** model, you must specify the following values:

  – **Critical We** is the critical Weber number in Equation 16.406 in the Theory Guide.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **Core B1** is B in Equation 16.407 in the Theory Guide.

– **Target Np** is the number of droplets given to each child parcel, before scaling is used to give the correct overall mass.

– **Xi** is $\langle \xi \rangle$ in Equation 16.408 in the Theory Guide.

> **Note**
>
> **Xi** is a negative value: exp(Xi) is a typical factor by which daughter particles are smaller than the original parcel.

For steady-state simulations, you also must specify an appropriate **Particle Time Step Size** and the **Number of Time Steps**, which will control the spray density. See Options for Interaction with the Continuous Phase (p. 1241) for more information.

Note that you may want to use the dynamic drag law when you use one of the breakup models. See Drag Laws (p. 1249) for information about choosing the drag law.

23.3.18. Specifying Turbulent Dispersion of Particles

As mentioned in Defining Injection Properties (p. 1286), you can choose for each injection stochastic tracking or cloud tracking as the method for modeling turbulent dispersion of particles.

23.3.18.1. Stochastic Tracking

For turbulent flows, if you choose to use the stochastic tracking technique, you must enable the **Discrete Random Walk Model** and specify the **Number of Tries**. Stochastic tracking includes the effect of turbulent velocity fluctuations on the particle trajectories using the DRW model described in Stochastic Tracking in the Theory Guide.

1. Click the **Turbulent Dispersion** tab in the **Set Injection Properties** dialog box.

2. Enable stochastic tracking by turning on the **Discrete Random Walk Model** under **Stochastic Tracking**.

3. Specify the **Number of Tries**:

   Selecting the **Turbulent Dispersion** model tells ANSYS Fluent to include turbulent velocity fluctuations in the particle force balance as in Equation 16.22 in the Theory Guide. The trajectory is computed more than once if your input exceeds 1: two trajectory calculations are performed if you enter 2, three trajectory calculations are performed if you enter 3, and so on. Each trajectory calculation includes a new stochastic representation of the turbulent contributions to the trajectory equation.

   When a sufficient number of tries is requested, the trajectories computed will include a statistical representation of the spread of the particle stream due to turbulence. Note that for unsteady particle tracking, the **Number of Tries** is set to 1 if using stochastic tracking.

If you want the characteristic lifetime of the eddy to be random (Equation 16.33 in the Theory Guide), enable the **Random Eddy Lifetime** option. You will generally not need to change the **Time Scale Constant** (C_L in Equation 16.24 in the Theory Guide) from its default value of 0.15, unless you are using the Reynolds Stress turbulence model (RSM), in which case a value of 0.3 is recommended.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.20: Mean Trajectory in a Turbulent Flow (p. 1295) illustrates a discrete phase trajectory calculation computed without turbulent dispersion and Figure 23.21: Stochastic Trajectories in a Turbulent Flow (p. 1295) illustrates the "stochastic" tracking (number of tries ≥ 1) option.

When multiple stochastic trajectory calculations are performed, the momentum and mass defined for the injection are divided evenly among the multiple particle/droplet tracks, and are therefore spread out in terms of the interphase momentum, heat, and mass transfer calculations. Including turbulent dispersion in your model can thus have a significant impact on the effect of the particles on the continuous phase when coupled calculations are performed.

Figure 23.20: Mean Trajectory in a Turbulent Flow

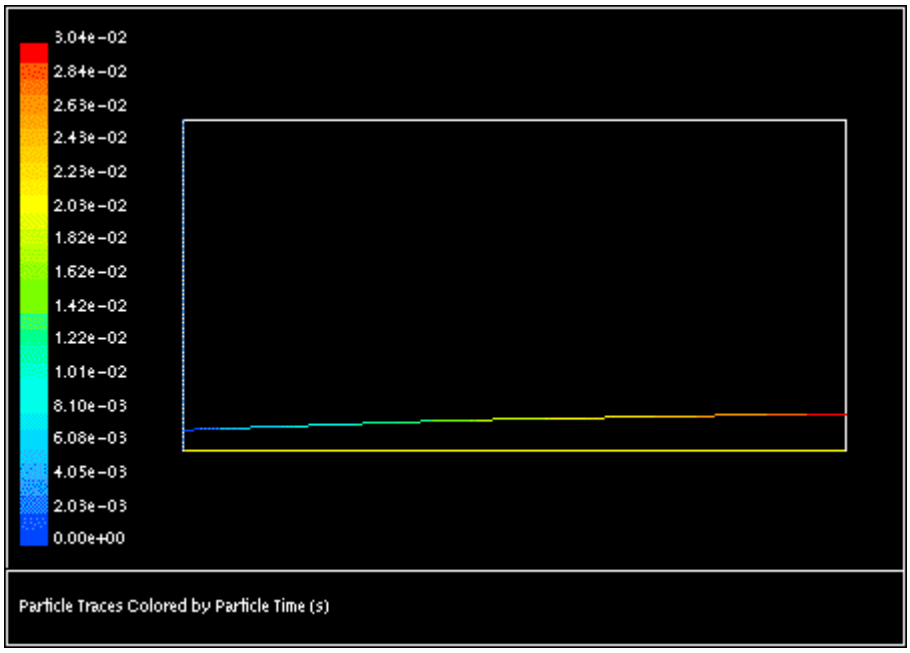
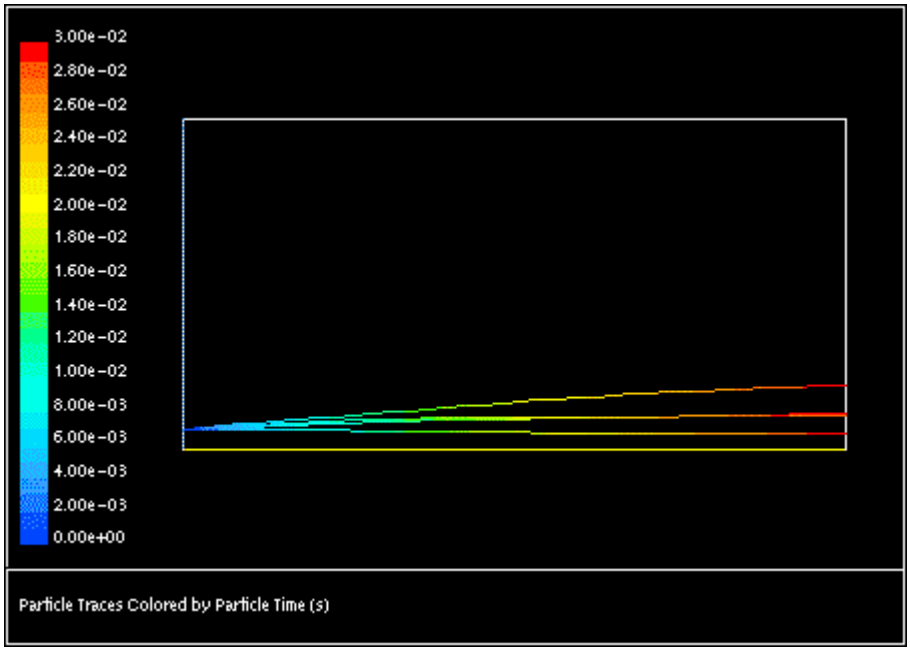


Figure 23.21: Stochastic Trajectories in a Turbulent Flow



23.3.18.2. Cloud Tracking

For turbulent flows, you can also include the effects of turbulent dispersion on the injection. Note that cloud tracking is not available for the massless particle type. When cloud tracking is used, the trajectory will be tracked as a cloud of particles about a mean trajectory, as described in Particle Cloud Tracking in the Theory Guide.

1. Click the **Turbulent Dispersion** tab in the **Set Injection Properties** dialog box.

2. Enable cloud tracking by turning on the **Cloud Model** under **Cloud Tracking**.

3. Specify the minimum and maximum cloud diameters. Particles enter the domain with an initial cloud diameter equal to the **Min. Cloud Diameter**. The particle cloud's maximum allowed diameter is specified by the **Max. Cloud Diameter**.

   You may want to restrict the **Max. Cloud Diameter** to a relevant length scale for the problem to improve computational efficiency in complex domains where the mean trajectory may become stuck in recirculation regions.

Important

Note that this model is available only in serial or for shared-memory tracking.

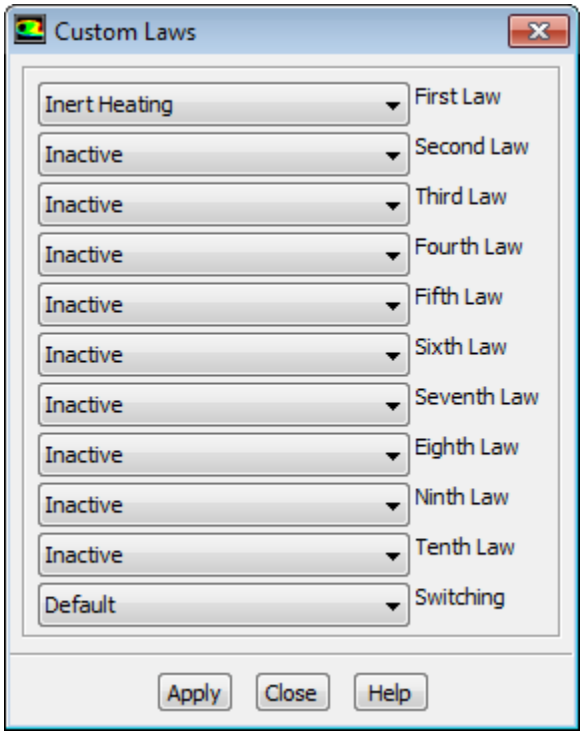
23.3.19. Custom Particle Laws

If the standard ANSYS Fluent laws, Laws 1 through 7, do not adequately describe the physics of your discrete phase model, you can modify them by creating custom laws with user-defined functions. More information about user-defined functions can be found in the Fluent Customization Manual. You can also create custom laws by using a subset of the existing ANSYS Fluent laws (for example, Laws 1, 2, and 4), or a combination of existing laws and user-defined functions.

Once you have defined and loaded your user-defined function(s), you can create a custom law by enabling the **Custom** option under **Laws** in the Set Injection Properties Dialog Box (p. 2796). This will open the Custom Laws Dialog Box (p. 2684). In the drop-down list to the left of each of the particle laws, you can select the appropriate particle law for your custom law. Each list contains the available options that can be chosen (the standard laws plus any user-defined functions you have loaded).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.22: The Custom Laws Dialog Box



There is a final drop-down list in the Custom Laws Dialog Box (p. 2684) labeled **Switching**. You may want to have ANSYS Fluent vary the laws used depending on conditions in the model. You can customize the way ANSYS Fluent switches between laws by selecting a user-defined function from this drop-down list (see `DEFINE_DPM_SWITCH` in the Fluent Customization Manual).

An example of when you might want to use a custom law might be to replace the standard devolatilization law with a specialized devolatilization law that more accurately describes some unique aspects of your model. After creating and loading a user-defined function that details the physics of your devolatilization law, you would visit the Custom Laws Dialog Box (p. 2684) and replace the standard devolatilization law (Law 2) with your user-defined function.

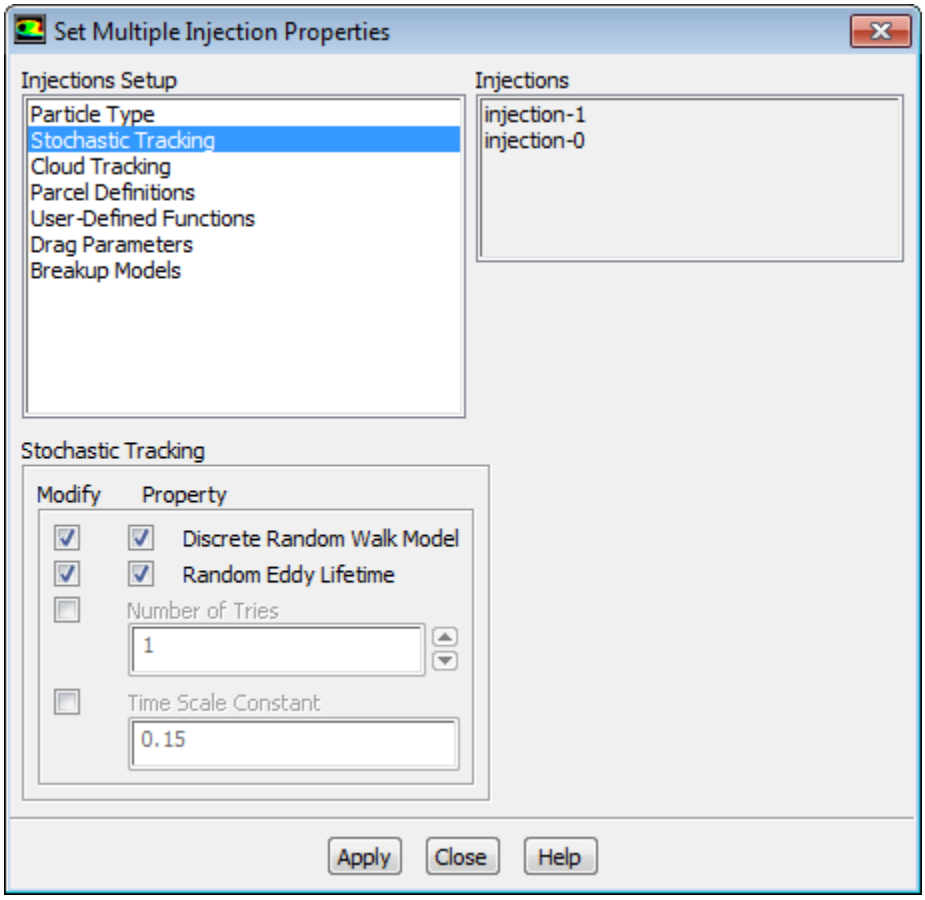
23.3.20. Defining Properties Common to More than One Injection

If you have a number of injections for which you want to set the same properties, ANSYS Fluent provides a shortcut so that you do not need to visit the **Set Injection Properties** dialog box for each injection to make the same changes.

As described in Modifying Injections (p. 1285), if you select more than one injection in the **Injections** dialog box, clicking the **Set...** button will open the **Set Multiple Injection Properties** dialog box (Figure 23.23: The Set Multiple Injection Properties Dialog Box (p. 1298)) instead of the **Set Injection Properties** dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.23: The Set Multiple Injection Properties Dialog Box



Depending on the type of injections you have selected (single, group, atomizers, and so on), there will be different categories of properties listed under **Injections Setup**. The names of these categories correspond to the headings within the **Set Injection Properties** dialog box (for example, **Particle Type** and **Stochastic Tracking**). Only those categories that are appropriate for all of your selected injections (which are shown in the **Injections** list) will be listed. If all of these injections are of the same type, more categories of properties will be available for you to modify. If the injections are of different types, you will have fewer categories to select from.

Note

You cannot define any properties as input parameters when you are modifying multiple injections. If you want to specify a property as an input parameter you must use Figure 23.18: The Set Injection Properties Dialog Box (p. 1287).

23.3.20.1. Modifying Properties

To modify a property, perform the following steps:

1. Select the appropriate category in the **Injections Setup** list. For example, if you want to set the same flow rate for all of the selected injections, select **Point Properties**. The dialog box will expand to show the properties that appear under that heading in the **Set Injection Properties** dialog box.

2. Set the property (or properties) to be modified, as described below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **Apply**. ANSYS Fluent will report the change in the console window.

Important

You must click **Apply** to save the property settings within each category. If, for example, you want to modify the flow rate and the stochastic tracking parameters, you will need to select **Point Properties** in the **Injections Setup** list, specify the flow rate, and click **Apply**. You would then repeat the process for the stochastic tracking parameters, clicking **Apply** again when you are done.

There are two types of properties that can be modified using the **Set Multiple Injection Properties** dialog box.

The first type involves one of the following actions:

- selecting a value from a drop-down list
- making a selection from a list of radio buttons

The second type involves one of the following actions:

- entering a value in a field
- enabling / disabling an option

Setting the first type of property works the same way as in the **Set Injection Properties** dialog box. For example, if you select **Particle Type** in the **Injections Setup** list, the dialog box will expand to show the portion of the **Set Injection Properties** dialog box where you choose the particle type. You can simply choose the desired type and click **Apply**.

Setting the second type of property requires an additional step. If you select a category in the **Injections Setup** list that contains this type of property, the expanded portion of the dialog box will look like the corresponding part of the **Set Injection Properties** dialog box, with the addition of **Modify** check buttons (see Figure 23.23: The Set Multiple Injection Properties Dialog Box (p. 1298)). To change one of the properties, first turn on the **Modify** check button to its left, and then specify the desired status or value.

For example, if you would like to enable stochastic tracking, first turn on the **Modify** check button to the left of **Stochastic Model**. This will make the property active so you can modify its status. Then, under **Property**, turn on the **Stochastic Model** check button. (Be sure to click **Apply** when you are done setting stochastic tracking parameters.)

If you would like to change the value of **Number of Tries**, select the **Modify** check button to its left to make it active, and then enter the new value in the field. Make sure you click **Apply** when you have finished modifying the stochastic tracking properties.

Important

The setting for a property that has not been enabled with the **Modify** check button is not relevant, because it will not be applied to the selected injections when you click **Apply**. After you enable **Modify** for a particular property, clicking **Apply** will modify that property for all of the selected injections, so make sure that you have the settings the way that you want

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

them before you do this. If you make a mistake, you will have to return to the **Set Injection Properties** dialog box for each injection to fix the incorrect setting, if it is not possible to do so in the **Set Multiple Injection Properties** dialog box.

23.3.20.2. Modifying Properties Common to a Subset of Selected Injections

Note that it is possible to change a property that is relevant for only a subset of the selected injections. For example, if some of the selected injections are using stochastic tracking and some are not, enabling the **Random Eddy Lifetime** option and clicking **Apply** will turn this option on only for those injections that are using stochastic tracking. The other injections will be unaffected.

23.3.21. Point Properties for Transient Injections

Simulations of transient particles often require time dependent injection conditions. Potentially most of the point properties may change over time. One method to accomplish this is the use of an **Initialization** function as described in `DEFINE_DPM_INJECTION_INIT` in the Fluent Customization Manual.

An easier way is to use transient profiles containing one or more variables based on the time or crank angle that can be assigned to various point properties: (Total) **Mass Flow Rate**, **X-, Y-, Z- Velocity**, **Velocity Magnitude,** and **Cone Angle** depending on the injection type selected. See Point Properties for Single Injections (p. 1269) to Point Properties for Effervescent Atomizer Injections (p. 1280) for the description of the individual properties.

Before transient profile variables can be assigned to point properties of injections, a profile file has to be read into ANSYS Fluent using the **File**/**Read**/**Profile...** ribbon tab item. In the **Select File** dialog box a **transient profile** in tabular format with extension `.ttab` has to be chosen. Alternatively, you can read this file into ANSYS Fluent using the `read-transient-table` text command.

`file` → `read-transient-table`

The profile name should not exceed 63 characters. See Defining Transient Cell Zone and Boundary Conditions (p. 448) for the format description. The following example illustrates an injection within 3 intervals of 2.5 milliseconds injection time, where the second injection has an elevated velocity.

```
mv-profile 3 13 0
time         mass-flow    velocity
0            0            0.1
0.00999      0            0.1
0.01         0.001        0.1
0.0125       0.001        0.1
0.01251      0            0.1
0.01999      0            0.1
0.02         0.001        5
0.0225       0.001        5
0.02251      0            0.1
0.02999      0            0.1
0.03         0.001        0.1
0.0325       0.001        0.1
0.03251      0            0.1
```

For the setting of point properties go to the **Point Properties** tab in the **Set Injection Properties** dialog box.

When transient profiles are loaded, you can choose either a **constant** method (the default) or **profile** from the **Method** drop-down list. If you select **profile**, you also must choose a profile from the corres-

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ponding drop-down list that appears in the **Value** column. The variable angle or time from the file is used as an independent interpolation variable and cannot be chosen as a profile.

Important

All quantities in the profile file, including coordinate values, must be specified in SI units. Specifically, Mass Flow Rate must be specified in kg/s, Velocity in m/s, and Cone Angle in radians. ANSYS Fluent does not perform unit conversion when reading profile files.

23.4. Setting Boundary Conditions for the Discrete Phase

When a particle reaches a physical boundary (for example, a wall or inlet boundary) in your model, ANSYS Fluent applies a discrete phase boundary condition to determine the fate of the trajectory at that boundary. One of several contingencies may arise:

- The particle may be reflected via an elastic or inelastic collision.
- The particle may escape through the boundary. The particle is lost from the calculation at the point where it impacts the boundary.
- The particle may be trapped at the wall. Nonvolatile material is lost from the calculation at the point of impact with the boundary; volatile material present in the particle or droplet is released to the vapor phase at this point.
- The particle may pass through an internal boundary zone, such as radiator or porous jump.
- The particle may slide along the wall, depending on particle properties and impact angle.
- The particle may form a film (wall film model).

You also have the option of implementing a user-defined function to model the particle behavior when hitting the boundary. More information about user-defined functions can be found in the Fluent Customization Manual.

The boundary condition, or trajectory fate, can be defined separately for each zone in your ANSYS Fluent model.

For additional information, see the following sections:

23.4.1. Discrete Phase Boundary Condition Types
23.4.2. Default Discrete Phase Boundary Conditions
23.4.3. Coefficients of Restitution
23.4.4. Friction Coefficient
23.4.5. Particle-Wall Impingement Heat Transfer
23.4.6. Film Condensation Model
23.4.7. Wall Boundary Layer Model
23.4.8. Setting Particle Erosion and Accretion Parameters

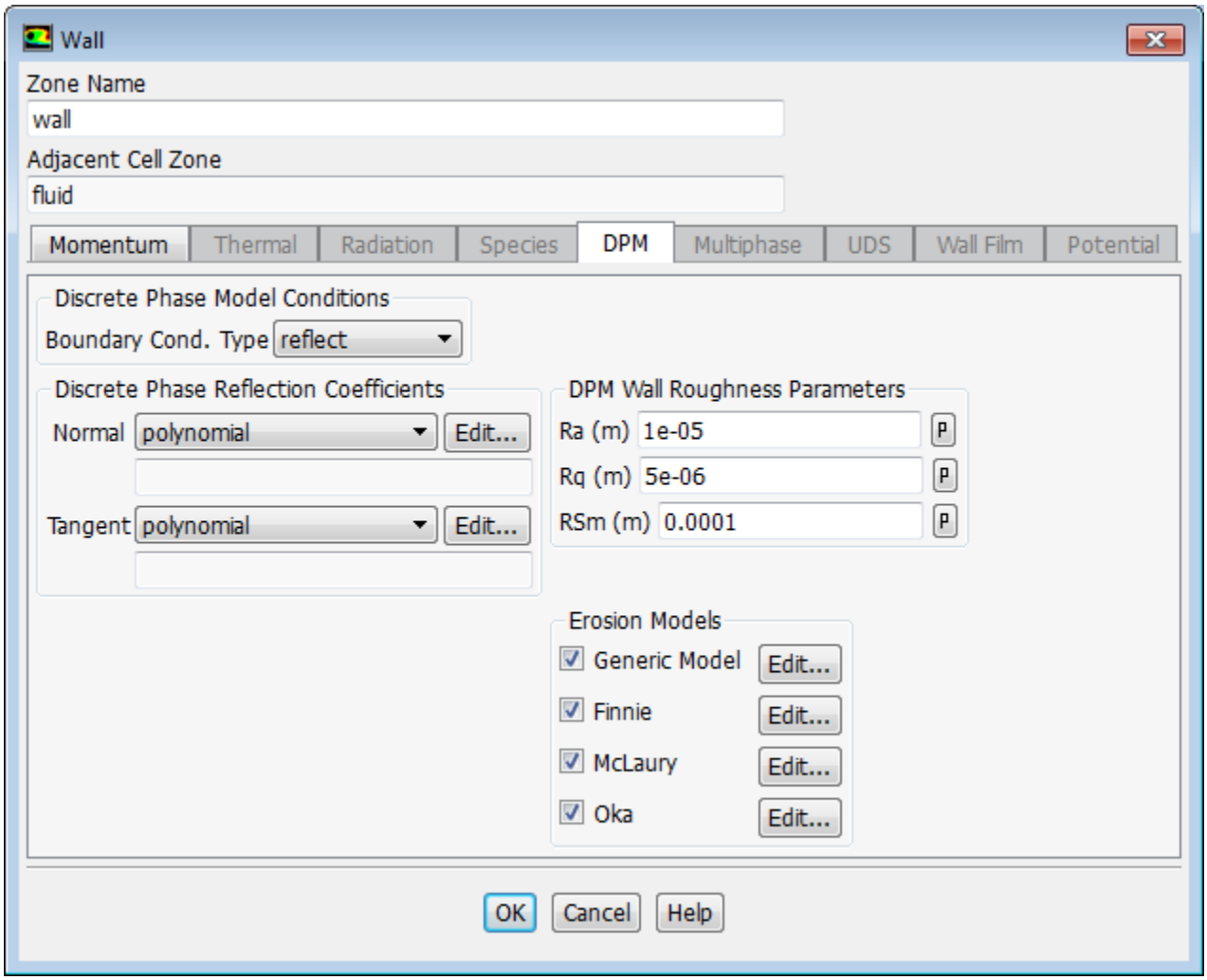
23.4.1. Discrete Phase Boundary Condition Types

Discrete phase boundary conditions can be set for boundaries in the dialog boxes opened from the **Boundary Conditions** task page. When one or more injections have been defined, inputs for the discrete phase will appear in the dialog boxes. In the **Walls** dialog boxes, you will need to click the **DPM** tab to

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

access the **Discrete Phase Model Conditions** as shown in Figure 23.24: Discrete Phase Boundary Conditions in the Wall Dialog Box (p. 1302).

Figure 23.24: Discrete Phase Boundary Conditions in the Wall Dialog Box



The available boundary conditions are:

- `reflect`

  The particle rebounds off the boundary with a change in its momentum. ANSYS Fluent provides two models for computing this change:

  – For non-rotating particles, the momentum depends on the coefficient of restitution (Equation 16.208 in the *Fluent Theory Guide*).

  – For rotating particles, the model proposed by Tsuji et al. 489 is used. The particle rotation model includes frictional effects.

  The details on these two models can be found in Wall-Particle Reflection Model Theory in the *Fluent Theory Guide*.

  For information on how to set up the parameters for non-rotating and rotating particle models, see Coefficients of Restitution (p. 1305) and Friction Coefficient (p. 1305), respectively.

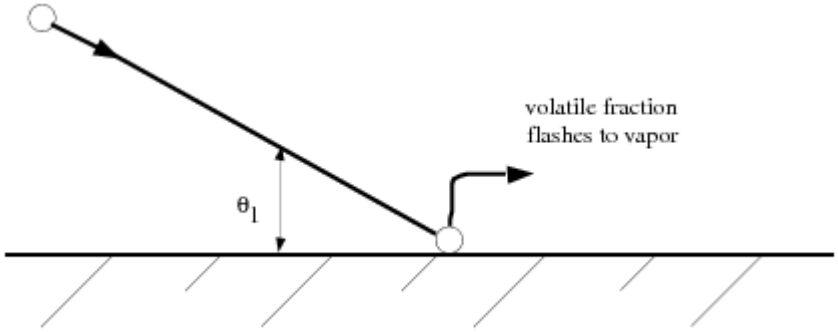
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When at least one injection accounts for the rough wall model (the **Rough Wall Model** option is selected under the **Physical Models** tab in the **Set Injections Properties** dialog box), you can specify the wall surface roughness parameters as described in Rough Wall Model (p. 1292).

- `trap`

  The trajectory calculations are terminated and the fate of the particle is recorded as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 23.25: "Trap" Boundary Condition for the Discrete Phase (p. 1303). In the case of combusting particles, the remaining volatile mass is passed into the vapor phase.

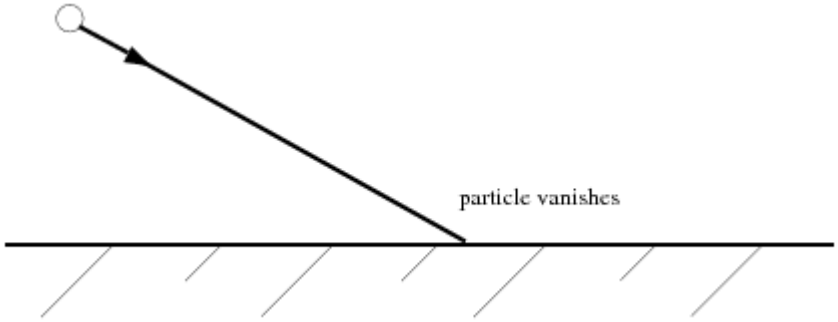
  Figure 23.25: "Trap" Boundary Condition for the Discrete Phase

- `escape`

  The particle is reported as having "escaped" when it encounters the boundary in question. Trajectory calculations are terminated. See Figure 23.26: "Escape" Boundary Condition for the Discrete Phase (p. 1303).

  Figure 23.26: "Escape" Boundary Condition for the Discrete Phase

- `wall-jet`

  The `wall-jet` type boundary condition is appropriate for high-temperature walls where no significant liquid film is formed, and in high-Weber-number impacts where the spray acts as a jet. The model is not appropriate for regimes where film is important (for example, port fuel injection in SI engines, rainwater runoff, and so on).

  A more detailed description of underlying theory is available in Wall-Jet Model Theory in the Theory Guide.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `wall-film`

  This boundary condition offers the following two impingement models:

  `stanton-rutland`: (default) consists of four regimes, namely, stick, rebound, spread, and splash, which are based on the impact energy and wall temperature. See The Stanton-Rutland Model in the *Fluent Theory Guide* for more details.

  `kuhnke`: consists of four regimes spread, rebound, splash, and dry splash (thermal breakup). See The Kuhnke Model in the *Fluent Theory Guide* for more details.

  For the `kuhnke` model, you must specify the following **DPM Wall Roughness Parameters**:

  Roughness Ra: is used to compute K_{cold} in Equation 16.258 in the *Fluent Theory Guide*.

  Roughness Rz: is used to compute splashed droplets ejection angle in Equation 16.270 in the *Fluent Theory Guide*.

  If you want to model splashing, you can specify **Number of Splash Drops**. A value of zero implies that the splashing model will not be used.

  The **Critical Temperature Factor** is used to determine the transition temperature (Equation 16.222 and Equation 16.250 in the *Fluent Theory Guide* for the `stanton-rutland` and `kuhnke` splashing models, respectively).

  Detailed information on the wall film model can be found in Wall-Film Model Theory in the *Fluent Theory Guide*.

  For a list of limitations that exist with `wall-film` boundary conditions, see Limitations on Using the Lagrangian Wall Film Model (p. 1240).

- `interior`

  This boundary condition means that the particles will pass through the internal boundary. This option is available only for internal boundary zones, such as a radiator or a porous jump.

It is also possible to use a user-defined function to compute the behavior of the particles at a physical boundary. More information about user-defined functions can be found in the Fluent Customization Manual.

Because you can stipulate any of these conditions at flow boundaries, it is possible to incorporate mixed discrete phase boundary conditions in your ANSYS Fluent model.

23.4.2. Default Discrete Phase Boundary Conditions

ANSYS Fluent makes the following assumptions regarding boundary conditions:

- The `reflect` type is assumed at wall, symmetry, and axis boundaries, with both coefficients of restitution equal to 1.0
- The `escape` type is assumed at all flow boundaries (pressure and velocity inlets, pressure outlets, and so on)
- The `interior` type is assumed at all internal boundaries (radiator, porous jump, and so on)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The coefficient of restitution can be modified only for wall boundaries.

23.4.3. Coefficients of Restitution

A normal or tangential coefficient of restitution equal to 1.0 implies that the particle retains all of its normal or tangential momentum after the rebound (an elastic collision). A normal or tangential coefficient of restitution equal to 0.0 implies that the particle retains none of its normal or tangential momentum after the rebound.

Nonconstant coefficients of restitution can be specified for wall zones with the `reflect` type boundary condition. The coefficients are set as a function of the impact angle, θ_1, in Particle Reflection at Wall in the *Fluent Theory Guide*.

Note that the default setting for both coefficients of restitution is a constant value of 1.0 (all normal and tangential momentum retained).

If you select the `reflect` type at a wall (only), you can define the **Normal** and **Tangent** coefficients of restitution under **Discrete Phase Reflection Coefficients**. You can select a `constant`, `polynomial`, `piecewise-linear`, or `piecewise-polynomial` function from the relevant drop-down lists. The dialog boxes for defining the polynomial, piecewise-linear, and piecewise-polynomial functions are the same as those used for defining temperature-dependent properties. The applied wall heat transfer model assumes that a liquid is getting in contact with the wall. See Defining Properties Using Temperature-Dependent Functions (p. 475) for details.

23.4.4. Friction Coefficient

When at least one injection accounts for particle rotation (the **Enable Rotation** option is selected under the **Physical Models** tab in the **Set Injections Properties** dialog box), you can specify friction coefficients for wall zones with the `reflect` type boundary condition. In the **Wall** dialog box, under the **DPM** tab, you can define the **Friction Coefficient** as a `constant` or as a `polynomial`, `piecewise-linear`, or `piecewise-polynomial` function.

The default value for the `constant` is 0.2. For the piecewise-linear and piecewise-polynomial profiles, the friction coefficient is computed in terms of the relative tangential velocity. The default inputs for these functions are derived from the friction collision law (The Friction Collision Law in the *Fluent Theory Guide*) and are as follows.

- The `piecewise-linear` approximation is defined using the following default points for the relative (particle-to-wall) tangential velocity:
  - Point 1: 0 m/s
  - Point 2: 1 m/s
  - Point 3: 10 m/s
  - Point 4: 210 m/s

- The `piecewise-polynomial` approximation is defined in the three relative (particle-to-wall) tangential velocity ranges as follows:
  - Range 1: 0–1 m/s: a quadratic function
  - Range 2: 1–10 m/s: a constant
  - Range 3: 10–210 m/s: a linear function

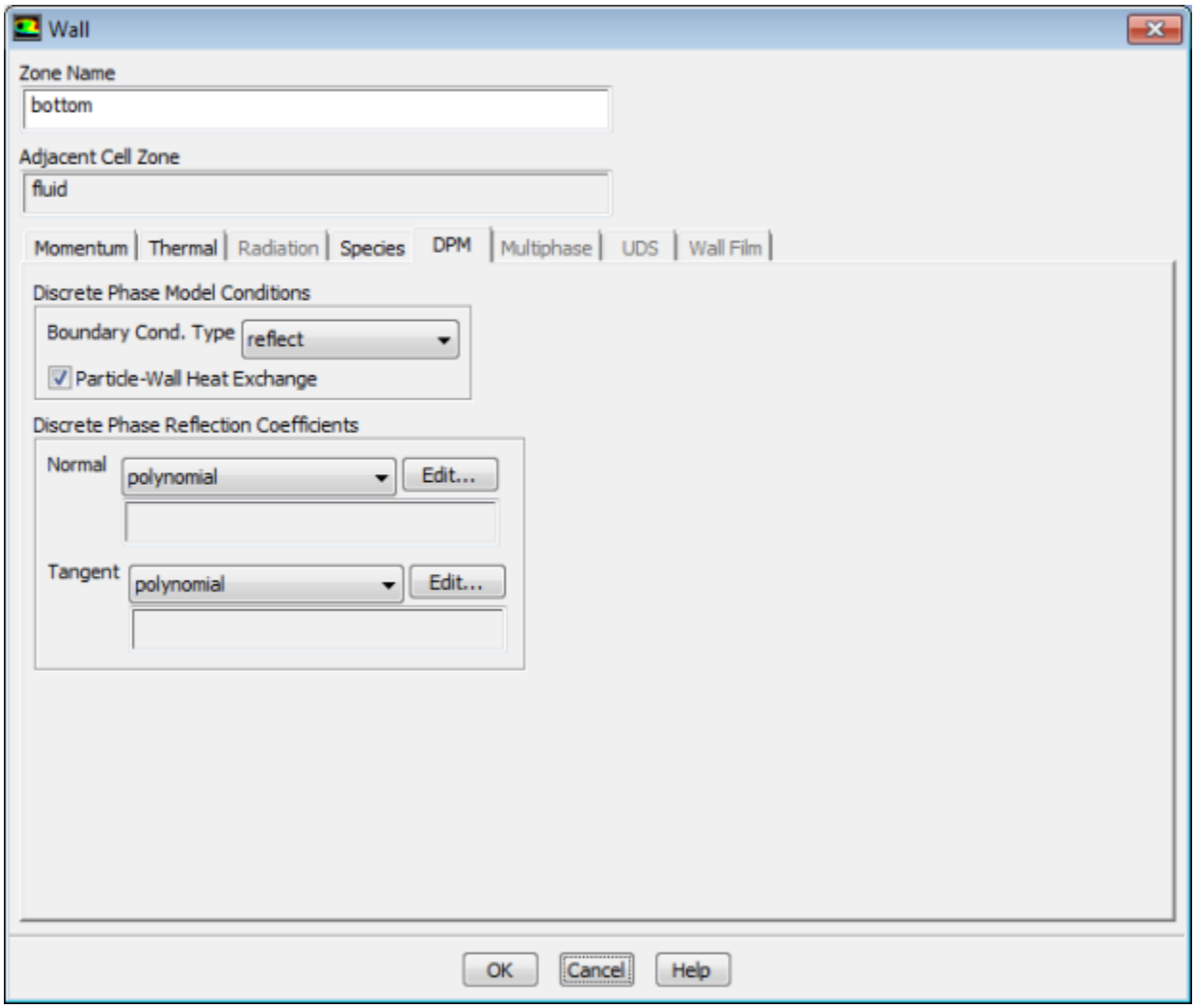
For details about using the **Piecewise-Linear Profile** dialog box and **Piecewise-Polynomial Profile** dialog box see Inputs for Piecewise-Linear Functions (p. 476) and Inputs for Piecewise-Polynomial Functions (p. 478), respectively.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23.4.5. Particle-Wall Impingement Heat Transfer

To enable the particle-to-wall heat exchange for the `reflect`, `wall-jet`, or `wall-film` boundary conditions: In the **Wall** dialog box, under the **DPM** tab, enable the **Particle-Wall Heat Exchange** option. The option is available only for wall boundary conditions and unsteady particle tracking when the energy equation is enabled.

Figure 23.27: The Wall Dialog Box: the Particle-Wall Heat Exchange Option



Note that when a particle is reflected from a wall with the wall film DPM boundary condition, the particle-to-wall heat exchange is calculated directly between the particle and the wall. Any wall film present is not taken into account.

The model is applied for all inert, droplet, and multicomponent particles impinging on the wall. The specific model is not applied for the splashed particles. When the wall film DPM boundary condition is active, the model is applied in the Rebound Regime, and it is assumed that the particles hit the wall irrespective of the presence of a film. In the Splashing regime, the mass fraction that is deposited mixes with any existing film, and the splashed particles retain the impinging droplet temperature.

For combusting particles, the wall heat transfer is calculated only if the **Wet Combustion Model** option is enabled in the **Set Injection Properties** dialog box, and the particle liquid fraction is nonzero, otherwise the **Particle-Wall Heat Exchange** option has no effect.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See Particle–Wall Impingement Heat Transfer Theory in the *Fluent Theory Guide* for the underlying theory and equations of the particle–wall impingement heat transfer model.

23.4.6. Film Condensation Model

Condensation can be defined as the transition from the gas phase to the liquid phase as a vapor condenses on a surface. The ANSYS Fluent film condensation model is triggered when the partial pressure of vapor species i in the bulk gas mixture exceeds either its vapor pressure at the wall-film or, in case no film exists, at the wall temperature. When these conditions are met, the vapor species will condense and form the liquid film on the wall faces.

The following limitations currently exist in the film condensation model:

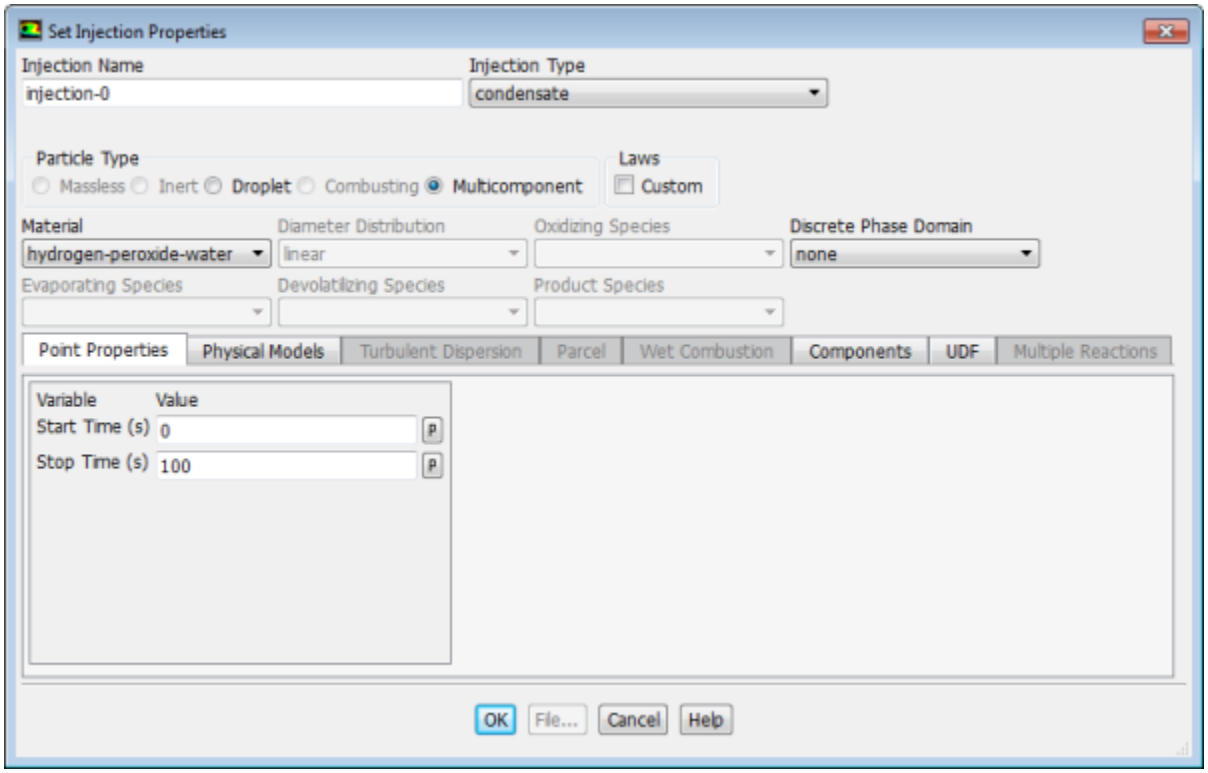
- The film particles are stagnant
- Only particles of a single material (droplet or multicomponent) can condense on a particular wall face
- At least one species in the mixture must be non-condensable

To use the film condensation model:

1. Enable the following models:
   - Energy equation
   - Species Transport model
   - **Unsteady Particle Tracking** (in the **Discrete Phase Model** dialog box)
2. If no droplet or multicomponent injection exists in the DPM case setup, create a condensate injection by selecting **condensate** from the **Injection Type** dropdown list in the **Set Injection Properties** dialog box and define the injection properties as appropriate.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.28: The Set Injection Properties Dialog Box: Condensate Injection



Note

A droplet or multicomponent injection must exist to define the vapor–liquid material pair for the condensation process.

For further information, see Injection Types (p. 1266), Point Properties for Condensate Injections (p. 1281), and Defining Injection Properties (p. 1286).

3. For each droplet and multicomponent injection, specify the condensable species by selecting the **Evaporating Species**.

4. In the **Wall** dialog box, under the **DPM** tab, select **wall-film** from the **Boundary Cond. Type** drop-down list and enable the **Film Condensation** option for each wall where condensation may occur. The Fluent solver automatically enables the **Wall Boundary Layer Model**. For more information, see Wall Boundary Layer Model (p. 1309).

Note

Film Condensation is available only with the **wall-film** DPM boundary condition.

5. Enable **Linearized Source Terms** in the **Numerics** tab of the **Discrete Phase Model** dialog box.

   Enabling the **Linearized Source Terms** option helps to stabilize the condensation solution and is strongly advisable.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the appropriate conditions exist, the evaporating species you have selected will condense on the walls that have the **Film Condensation** option enabled forming either droplet or multicomponent film particles (depending on the **Particle Type** selected in the **Set Injection Properties** dialog box).

For information about the theory behind the Film Condensation model, see Film Condensation in the *Fluent Theory Guide*.

23.4.7. Wall Boundary Layer Model

The Wall Boundary Layer model enables the boundary layer formulations for the mass and heat transfer equations for film vaporization and boiling as described in Film Vaporization and Boiling and Energy Transfer from the Film in the *Fluent Theory Guide*.

To enable the model, select **Wall Boundary Layer Model** in the **DPM** tab of the **Wall** dialog box. Note that the boundary layer formulations are always used for both condensation and vaporization rates when the Film Condensation model is enabled.

23.4.8. Setting Particle Erosion and Accretion Parameters

If the **Erosion/Accretion** option is selected in the **Physical Models** tab of the Discrete Phase Model Dialog Box (p. 2278) (see Monitoring Erosion/Accretion of Particles at Walls (p. 1251)), you can enable erosion models in the **DPM** tab of the Wall Dialog Box (p. 2456) and specify the corresponding erosion rate parameters at the walls.

Important

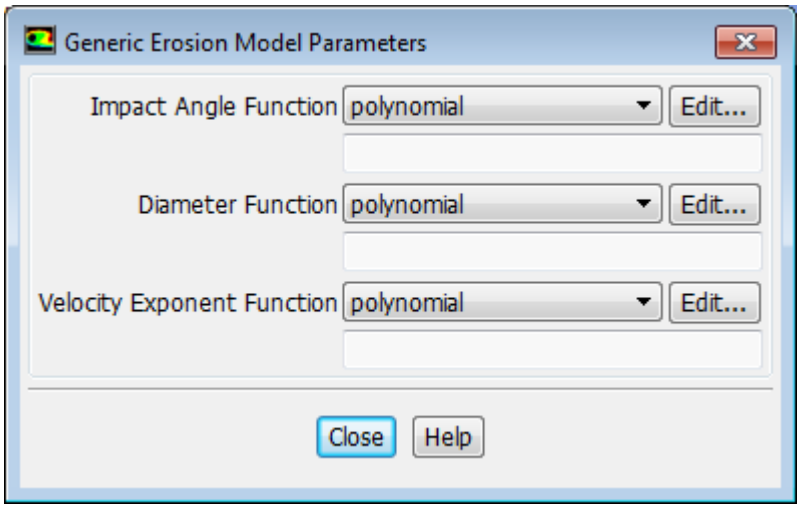
The default erosion rate parameters for each erosion model may not be appropriate for your analysis and must be adjusted based on your own experimental data.

The following models are available in ANSYS Fluent:

- **Generic Model**

  The erosion rate of the generic model is defined in Equation 16.308 in the Theory Guide as a product of the mass flux and specified functions for the particle diameter, impact angle, and velocity exponent. To specify the erosion parameters for the Generic model, click **Edit...** next to **Generic Model**.

Figure 23.29: TheGeneric Erosion Model Parameters Dialog Box



In the **Generic Erosion Model Parameters** dialog box, specify the following parameters using a `constant`, `polynomial`, `piecewise-linear`, or `piecewise-polynomial` function:

– **Impact Angle Function**: $f(\gamma)$ in Equation 16.308 in the *Fluent Theory Guide*.

– **Diameter Function**: $C(d_p)$ in Equation 16.308 in the *Fluent Theory Guide*.

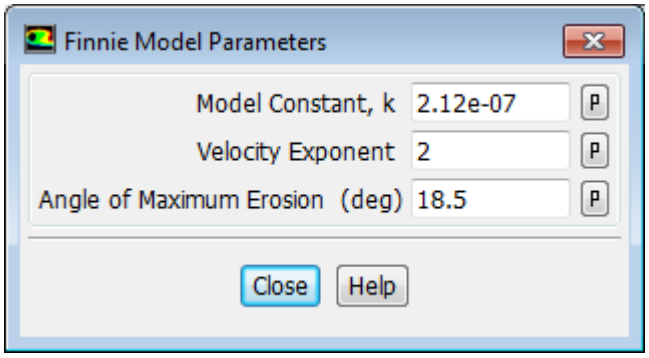
– **Velocity Exponent Function**: $b(v)$ in Equation 16.308 in the *Fluent Theory Guide*.

See Accretion in the Theory Guide for a detailed description of these functions and Defining Properties Using Temperature-Dependent Functions (p. 475) for details about using the dialog boxes for defining polynomial, piecewise-linear, and piecewise-polynomial functions.

- **Finnie**

  To specify the erosion parameters for the Finnie model, click **Edit...** next to **Finnie**.

  Figure 23.30: TheFinnie Model Parameters Dialog Box

  In the **Finnie Model Parameters** dialog box, specify the following parameters:

  – **Model Constant, k**: k in Equation 16.301 in the *Fluent Theory Guide*.

  – **Velocity Exponent**: n in Equation 16.301 in the *Fluent Theory Guide*. The default value is 2. The value of the **Velocity Exponent** is generally within the range of 2.3 to 2.5 for metals.

  – **Angle of Maximum Erosion**: the impact angle between the approaching particle track and the wall at which the erosion reaches maximum.

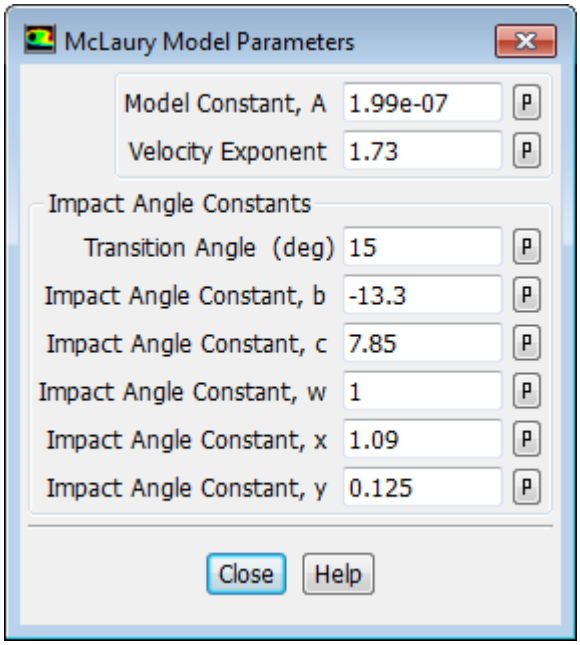
    For information about the theory behind the Finnie model, see Finnie Erosion Model in the *Fluent Theory Guide*.

- **McLaury**

  To specify the erosion parameters for the McLaury model, click **Edit...** next to **McLaury**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.31: TheMcLaury Model Parameters Dialog Box



In the **McLaury Model Parameters** dialog box, specify the following parameters:

– **Model Constant, A**: A in Equation 16.305 in the *Fluent Theory Guide*.

– **Velocity Exponent**: V in Equation 16.305 in the *Fluent Theory Guide*.

– **Transition Angle**: $\gamma_{\lim}$ in Equation 16.306 and Equation 16.307 in the *Fluent Theory Guide*.

– **Impact Angle Constant, b**: b in Equation 16.306 in the *Fluent Theory Guide*.

– **Impact Angle Constant, c**: c in Equation 16.306 in the *Fluent Theory Guide*.

– **Impact Angle Constant, w**: w in Equation 16.307 in the *Fluent Theory Guide*.

– **Impact Angle Constant, x**: x in Equation 16.307 in the *Fluent Theory Guide*.

– **Impact Angle Constant, y**: y in Equation 16.307 in the *Fluent Theory Guide*.

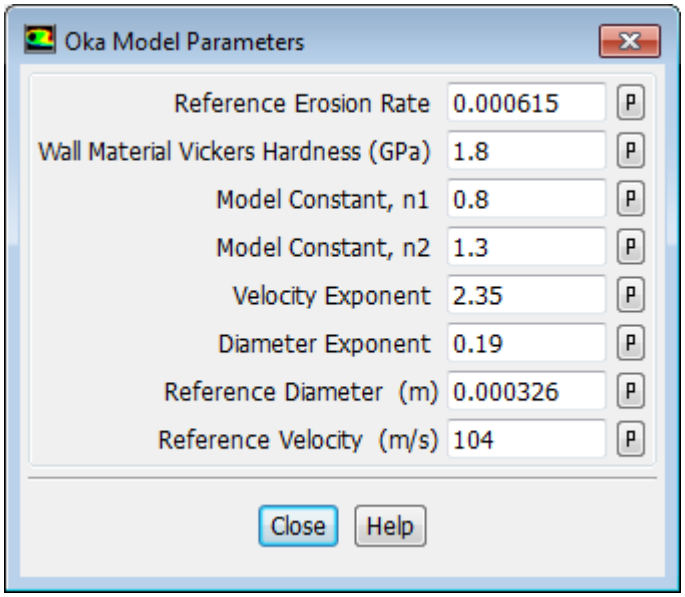
See Example of the McLaury Erosion Model Constants in the *Fluent Theory Guide* for more information about the McLaury model parameters. For theoretical background on the McLaury model, see McLaury Erosion Model in the *Fluent Theory Guide*.

- **Oka**

  To specify the erosion parameters for the Oka model, click **Edit...** next to **Oka**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.32: TheOka Model Parameters Dialog Box



In the **Oka Model Parameters** dialog box, specify the following parameters:

– **Reference Erosion Rate**: E_{90} in Equation 16.303 in the *Fluent Theory Guide*.

– **Wall Material Hardness**: H_v in Equation 16.304 in the *Fluent Theory Guide*.

– **Model Constant, n1**: n_1 in Equation 16.304 in the *Fluent Theory Guide*.

– **Model Constant, n2**: n_2 in Equation 16.304 in the *Fluent Theory Guide*.

– **Velocity Exponent**: k_2 in Equation 16.303 in the *Fluent Theory Guide*.

– **Diameter Exponent**: k_3 in Equation 16.303 in the *Fluent Theory Guide*.

– **Reference Diameter**: d_{ref} in Equation 16.303 in the *Fluent Theory Guide*.

– **Reference Velocity**: V_{ref} in Equation 16.303 in the *Fluent Theory Guide*.

See Example of the Oka Erosion Model Constants in the *Fluent Theory Guide* for an example of the Oka model parameters' values. For more information on the theory behind the Oka model, see Oka Erosion Model in the *Fluent Theory Guide*.

By default, all erosion models are enabled.

Important

- The default constants for the Finnie, Oka, and McLaury models have been tuned to match the experiment described by Oka [75] (p. 2891) in which 326-micron sand particles impact a carbon steel wall at a speed of 104 m/s and therefore are only valid for those specific conditions. You can use the default values as a starting point for your simulation and then adjust them as needed.
- Predictive capabilities of the erosion models are limited and strongly case dependent.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23.5. Setting Material Properties for the Discrete Phase

In order to apply the physical models described in earlier sections to the prediction of the discrete phase trajectories and heat/mass transfer, ANSYS Fluent requires many physical property inputs.

For additional information, see the following sections:

23.5.1. Summary of Property Inputs
23.5.2. Setting Discrete-Phase Physical Properties

23.5.1. Summary of Property Inputs

Table 23.1: Property Inputs for Inert Particles (p. 1313) – Table 23.5: Property Inputs for Multicomponent Particles (Law 7) (p. 1315) summarize which of these property inputs are used for each particle type and in which of the equations for heat and mass transfer each property input is used. Detailed descriptions of each input are provided in Setting Discrete-Phase Physical Properties (p. 1316).

Table 23.1: Property Inputs for Inert Particles

Property	Symbol
density	ρ_p in Equation 16.1 in the Theory Guide
specific heat	c_p in Equation 16.92 and Equation 16.293
thermal conductivity	k_p in Equation 16.12 and k_l in Equation 16.294
particle emissivity	ε_p in Equation 16.92
particle scattering factor	f in Equation 5.34
thermophoretic coefficient	$D_{T,p}$ in Equation 16.11

Table 23.2: Property Inputs for Droplet Particles

Properties	Symbol
density	ρ_p in Equation 16.1 in the Theory Guide
specific heat	c_p in Equation 16.112 and Equation 16.293
thermal conductivity	k_p in Equation 16.12 and k_l in Equation 16.294
viscosity	μ in Equation 16.364
latent heat	h_{fg} in Equation 16.112, Equation 16.285, Equation 16.286, and Equation 16.300
vaporization temperature	T_{vap} in Equation 16.99
boiling point	T_{bp} in Equation 16.99, Equation 16.118
volatile component fraction	f_{v0} in Equation 16.100, Equation 16.119
binary diffusivity	$D_{i,m}$ in Equation 16.104 and D_i in Equation 16.279
diffusivity reference pressure	P_{ref} in Equation 16.110 in the *Fluent Theory Guide*
saturation vapor pressure	$p_{sat}(T)$ in Equation 16.102
heat of pyrolysis	h_{pyrol} in Equation 16.436

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Properties	Symbol
droplet surface tension	σ in Equation 16.333, Equation 16.363
particle emissivity	ε_p in Equation 16.112, Equation 16.123
particle scattering factor	f in Equation 5.34
composition averaging coefficient	α_Y in Equation 16.201
temperature averaging coefficient	α_T in Equation 16.200

Table 23.3: Property Inputs for Combusting Particles (Laws 1–4)

Properties	Symbol
density	ρ_p in Equation 16.1 in the Theory Guide
specific heat	c_p in Equation 16.92
thermal conductivity	k_p in Equation 16.12
latent heat	H_{lat} in Equation 16.438
vaporization temperature	T_{vap}=T_{bp} in Equation 16.124
volatile component fraction	f_{v0} in Equation 16.125
swelling coefficient	C_{sw} in Equation 16.157
burnout stoichiometric ratio	S_b in Equation 16.164
combustible fraction	f_{comb} in Equation 16.163
heat of reaction for burnout	H_{reac} in Equation 16.178
fraction of reaction heat given to solid	f_h in Equation 16.178
particle emissivity	ε_p in Equation 16.158, Equation 16.178
particle scattering factor	f in Equation 5.34
thermophoretic coefficient	$D_{T,p}$ in Equation 16.11
devolatilization model	
– law 4, constant rate	
– – constant	A_0 in Equation 16.126
– law 4, single rate	
– – pre-exponential factor	A_1 in Equation 16.127
– – activation energy	E in Equation 16.127
– law 4, two rates	
– – pre-exponential factors	A_1,A_2 in Equation 16.130, Equation 16.131
– – activation energies	E_1,E_2 in Equation 16.130, Equation 16.131
– – weighting factors	α_1,α_2 in Equation 16.132
– law 4, CPD	
– – initial fraction of bridges in coal lattice	p_0 in Equation 16.143

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Properties	Symbol
– – initial fraction of char bridges	c_0 in Equation 16.142
– – lattice coordination number	$\sigma+1$ in Equation 16.154
– – cluster molecular weight	$M_{w,1}$ in Equation 16.154
– – side chain molecular weight	$M_{w,\delta}$ in

Table 23.4: Property Inputs for Combusting Particles (Law 5)

Properties	Symbol
combustion model	
– law 5, diffusion rate	
– – binary diffusivity	$D_{i,m}$ in Equation 16.165 in the Theory Guide
– – diffusivity reference pressure	P_{ref} in Equation 16.110 in the *Fluent Theory Guide*
– law 5, diffusion/kinetic rate	
– – mass diffusion limited rate constant	C_1 in Equation 16.166
– – kinetics limited rate pre-exp. factor	C_2 in Equation 16.167
– – kinetics limited rate activation energy	E in Equation 16.167
– law 5, intrinsic rate	
– – mass diffusion limited rate constant	C_1 in Equation 16.166
– – kinetics limited rate pre-exp. factor	A_i in Equation 16.176
– – kinetics limited rate active energy	E_i in Equation 16.176
– – char porosity	θ in Equation 16.173
– – mean pore radius	$\overline{r}_p$ in Equation 16.175
– – specific internal surface area	A_g in Equation 16.170, Equation 16.172
– – tortuosity	τ in Equation 16.173
– – burning mode	α in Equation 16.177

Table 23.5: Property Inputs for Multicomponent Particles (Law 7)

Property	Symbol
mixture species	selected droplets for components
density	ρ_p in Equation 16.1 of the Theory Guide
specific heat	c_p in Equation 16.183 and Equation 16.293
thermal conductivity	k_p in Equation 16.12 and k_l in Equation 16.294
vapor particle equilibrium	$C_{i,s}$ in Equation 16.181
thermophoretic coefficient	$D_{T,p}$ in Equation 16.11
composition averaging coefficient	α_Y in Equation 16.201
temperature averaging coefficient	α_T in Equation 16.200

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23.5.2. Setting Discrete-Phase Physical Properties

23.5.2.1. The Concept of Discrete-Phase Materials

When you create a particle injection and define the initial conditions for the discrete phase (as described in Setting Initial Conditions for the Discrete Phase (p. 1265)), you choose a particular material as the particle's material. All particle streams of that material will have the same physical properties.

Important

Note that you will not choose a **Material** for a **Massless** particle type in the **Set Injections Properties** dialog box.

Discrete-phase materials are divided into four categories, corresponding to the four types of particles available. These material types are **inert-particle**, **droplet-particle**, **combusting-particle**, and **multicomponent-particle**. Each material type will be added to the **Material Type** list in the Create/Edit Materials Dialog Box (p. 2302) when an injection of that type of particle is defined (in the **Set Injection Properties** or **Set Multiple Injection Properties** dialog box, as described in Setting Initial Conditions for the Discrete Phase (p. 1265)). The first time you create an injection of each particle type, you will be able to choose a material from the database, and this will become the default material for that type of particle. That is, if you create another injection of the same type of particle, your selected material will be used for that injection as well. You may choose to modify the predefined properties for your selected particle material, if you want (as described in Modifying Properties of an Existing Material (p. 463)). If you need only one set of properties for each type of particle, you need not define any new materials; you can simply use the same material for all particles.

Important

If you do not find the material you want in the database, you can select a material that is close to the one you want to use, and then modify the properties and give the material a new name, as described in Creating a New Material (p. 466).

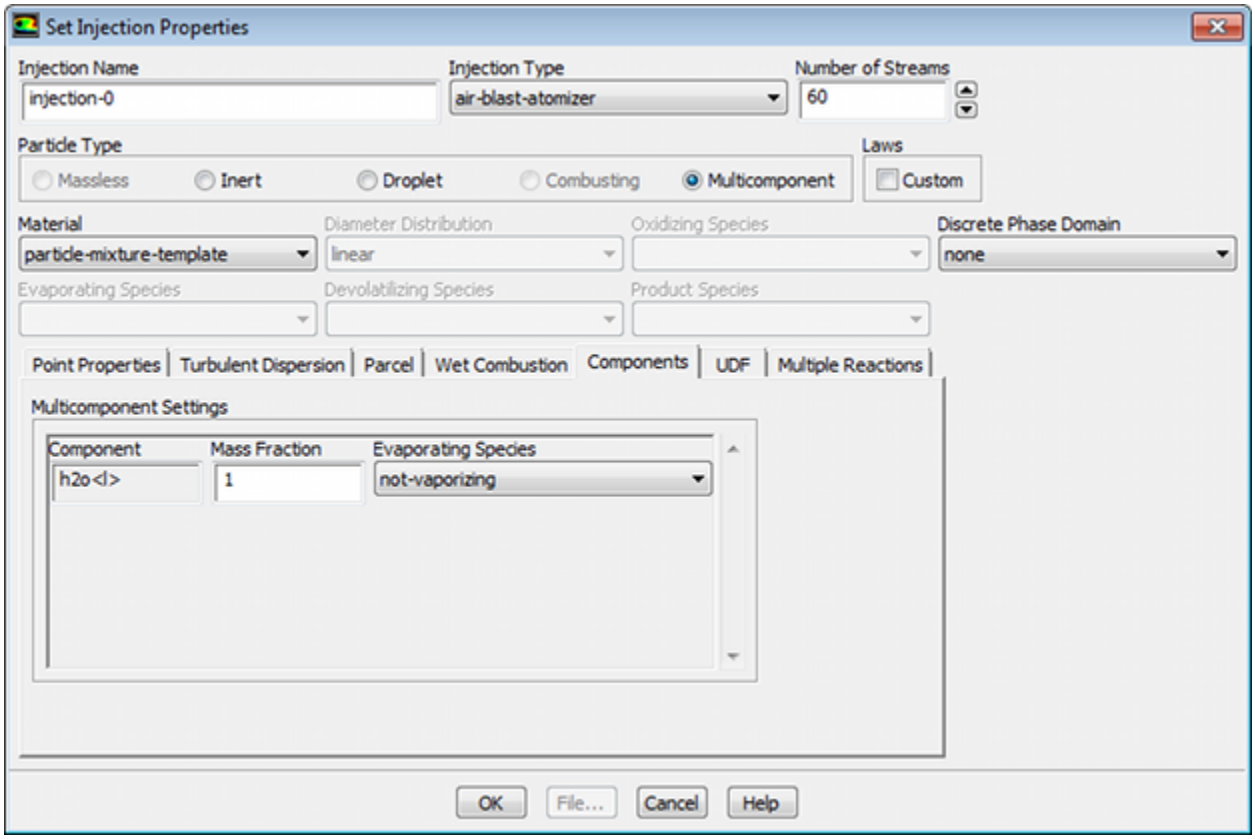
Important

Note that a discrete-phase material type will not appear in the **Material Type** list in the **Create/Edit Materials** dialog boxes until you have defined an injection of that type of particles. This means, for example, that you cannot define or modify any combusting-particle materials until you have defined a combusting particle injection (as described in Setting Initial Conditions for the Discrete Phase (p. 1265)).

For a **particle-mixture** material type, you must select the species in your mixture. To do this, click the **Edit...** button next to **Mixture Species** in the **Create/Edit Materials** dialog box. The **Species** dialog box will open, where you will include your **Selected Species**. The selected species will now be available in the **Set Injection Properties** dialog box, under the **Components** tab (Figure 23.33: The Components Tab (p. 1317)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.33: The Components Tab



23.5.2.1.1. Defining Additional Discrete-Phase Materials

In many cases, a single set of physical properties (density, heat capacity, and so on) is appropriate for each type of discrete phase particle considered in a given model. Sometimes, however, a single model may contain two different types of inert, droplet, combusting particles, or multicomponent particles (for example, heavy particles and gaseous bubbles or two different types of evaporating liquid droplets). In such cases, it is necessary to assign a different set of properties to the two (or more) different types of particles. This is easily accomplished by defining two or more inert, droplet, or combusting particle materials and using the appropriate one for each particle injection.

You can define additional discrete-phase materials either by copying them from the database or by creating them from scratch. See Using the **Create/Edit Materials** Dialog Box (p. 461) for instructions on using the Create/Edit Materials Dialog Box (p. 2302) to perform these actions.

Important

Recall that you must define at least one injection (as described in Setting Initial Conditions for the Discrete Phase (p. 1265)) containing particles of a certain type before you will be able to define additional materials for that particle type.

23.5.2.2. Description of the Properties

The properties that appear in the Create/Edit Materials Dialog Box (p. 2302) vary depending on the particle type (selected in the **Set Injection Properties** or **Set Multiple Injection Properties** dialog box, as described in Defining Injection Properties (p. 1286) and Defining Properties Common to More than One Injection (p. 1297)) and the physical models you are using in conjunction with the discrete-phase model.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

All properties you may need to define for a discrete-phase material are listed below (alphabetically). See Table 23.1: Property Inputs for Inert Particles (p. 1313) – Table 23.4: Property Inputs for Combusting Particles (Law 5) (p. 1315) to see which properties are defined for each type of particle.

Binary Diffusivity

is the mass diffusion coefficient used in the vaporization law (Law 2), $D_{i,m}$ in Equation 16.104 and D_i in Equation 16.279 in the Theory Guide. This input is also used to define the mass diffusion of the oxidizing species to the surface of a combusting particle, $D_{i,m}$, as given in Equation 16.165 in the Theory Guide. (Note that the diffusion coefficient inputs that you supply for the continuous phase are not used for the discrete phase.)

For **Droplet Particle** type **Materials**, select **film-averaged** from the **Binary Diffusivity** drop-down list and specify parameters for the film-averaged model (see Equation 16.108 in the Theory Guide) using the **Film Binary Diffusivity** dialog box. The **film-averaged** model is recommended when accurate temperature-dependent binary diffusivity data are available.

To apply the unity Lewis number model (Equation 16.109 in the *Fluent Theory Guide*) select **unity-lewis-number** from the **Binary Diffusivity** drop-down. The unity Lewis number model is a simplified approach and is not appropriate when the molecular weights of the evaporating species (or oxidizing species for combusting particles) and the gas-phase mixture are very different. The model can be applied, for example, for the evaporation of water, light hydrocarbons, or methanol in air, but is not appropriate for heavy hydrocarbons.

You also have the option of implementing a user-defined function to model the particle binary diffusivity. See `DEFINE_DPM_PROPERTY` in the *Fluent Customization Manual* for more information.

To enable the pressure dependence for the binary diffusivity, in the **Create/Edit Materials** dialog box, under **Properties**, from the **Diffusivity Reference Pressure** drop-down list, select `constant` (see Equation 16.110 in the *Fluent Theory Guide*) and enter a value for the reference pressure P_{ref} that corresponds to the value defined for the **Binary Diffusivity** material property. The pressure dependence option is available with all property methods for the binary diffusivity, including `user-defined`, with the exception of the unity Lewis number model, for which the pressure dependence is already accounted for through the density variable in the model equation.

Boiling Point

is the temperature, T_{bp}, at which the calculation of the boiling rate equation (Equation 16.120 in the Theory Guide) is initiated by ANSYS Fluent. When a droplet particle reaches the boiling point, ANSYS Fluent applies Law 3 and assumes that the droplet temperature is constant at T_{bp}. The boiling point denotes the temperature at which the particle law transitions from the vaporization law to the boiling law.

For multicomponent particles the boiling point of the components is used only as a reference temperature of the latent heat. Instead, the boiling starts when the sum of the partial component saturation pressures reach the total fluid pressure. The definition of the saturation pressure curve is therefore essential for the boiling of multicomponent particles.

You also have the option of implementing a user-defined function to model the particle boiling point. See `DEFINE_DPM_PROPERTY` in the *Fluent Customization Manual* for more information.

Burnout Stoichiometric Ratio

is the stoichiometric requirement, S_b, for the burnout reaction, Equation 16.164 in the Theory Guide, in terms of mass of oxidant per mass of char in the particle.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Combustible Fraction

is the mass fraction of char, f_{comb}, in the coal particle, that is, the fraction of the initial combusting particle that will react in the surface reaction, Law 5 (Equation 16.163 in the Theory Guide).

Combustion Model

defines which version of the surface char combustion law (Law5) is being used. If you want to use the default diffusion-limited rate model, retain the selection of **diffusion-limited** in the drop-down list to the right of **Combustion Model**. No additional inputs are necessary, because the binary diffusivity defined above will be used in Equation 16.165 in the Theory Guide.

To use the kinetics/diffusion-limited rate model for the surface combustion model, select **kinetics/diffusion-limited** in the drop-down list. The Kinetics/Diffusion-Limited Combustion Model Dialog Box (p. 2352) will appear and you will enter the **Mass Diffusion Limited Rate Constant** (C_1 in Equation 16.166 in the Theory Guide), **Kinetics Limited Rate Pre-exponential Factor** (C_2 in Equation 16.167), and **Kinetics Limited Rate Activation Energy** (E in Equation 16.167).

Note that the **Kinetics/Diffusion-Limited Combustion Model** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.

To use the intrinsic model for the surface combustion model, select **intrinsic-model** in the drop-down list. The Intrinsic Combustion Model Dialog Box (p. 2353) will appear and you will enter the **Mass Diffusion Limited Rate Constant** (C_1 in Equation 16.166 in the Theory Guide), **Kinetics Limited Rate Pre-exponential Factor** (A_i in Equation 16.176), **Kinetics Limited Rate Activation Energy** (E_i in Equation 16.176), **Char Porosity** (θ in Equation 16.173), **Mean Pore Radius** ($\overline{r}_p$ in Equation 16.175), **Specific Internal Surface Area** (A_g in Equation 16.170 Equation 16.172), **Tortuosity** (τ in Equation 16.173), and **Burning Mode, alpha** (α in Equation 16.177).

Note that the **Intrinsic Combustion Model** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.

To use the multiple surface reactions model, select **multiple-surface-reactions** in the drop-down list. ANSYS Fluent will display the Multiple Surface Reactions Dialog Box (p. 2354) informing you to open the **Reactions** dialog box, where you can review or modify the particle surface reactions that you specified as described in Overview of User Inputs for Modeling Species Transport and Reactions (p. 958). In addition, you can set property options for the char specific heat and density:

Composition Dependent Specific Heat

if this option is enabled, your input for the particle c_p property will be used to determine the specific heat of the volatiles and ash component. The specific heat of the char will be calculated as a mass-weighted average of the particle specific heat values, and the particle specific heat is finally calculated as the mass average of the char and volatiles+ash fractions. The specific heat of the particle surface species should be defined in the corresponding fluid materials of the Mixture material.

Composition Dependent Density

if this option is enabled, your input for the particle Density property will be used to determine the density of the volatiles and ash component. The density of the char will be calculated as a volume-weighted average value of the particle surface species densities, and the particle density is finally calculated as the volume-weighted average of the char and volatiles+ash fractions. The

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

density of the particle surface species should be defined in the corresponding fluid materials of the Mixture material.

Important

If you have not yet defined any particle surface reactions, you must be sure to define them now. See Using the Multiple Surface Reactions Model for Discrete-Phase Particle Combustion (p. 1002) for more information about using the multiple surface reactions model.

You will notice that the **Burnout Stoichiometric Ratio** and **Heat of Reaction for Burnout** are no longer available in the **Create/Edit Materials** dialog box, as these parameters are now computed from the particle surface reactions you defined in the **Reactions** dialog box.

Note that the multiple surface reactions model is available only if the **Particle Surface** option for **Reactions** is enabled in the **Species Model** dialog box. See User Inputs for Particle Surface Reactions (p. 1001) for details.

Composition Averaging Coefficient,

is the coefficient α_Y in Equation 16.201 in the *Fluent Theory Guide*. To assume bulk gas composition for the physical properties in the particle vaporization and heating rates equations, select `none` from the **Composition Averaging Coefficient** drop-down in the **Create/Edit Materials** dialog box, under **Properties**. To enable property averaging, select `constant` and enter a value between 0 and 1 for the **Composition Averaging Coefficient**. $\alpha_Y = 1$ corresponds to bulk gas mixture. For $\alpha_Y = 0$, the composition reverts to the particle surface composition.

Important

When you define the **Composition Averaging Coefficient** with a `constant` value other than 1, you must also specify **Specific Heat**, **Viscosity** and **Thermal Conductivity** for the evaporating species (in the **Create/Edit Materials** dialog box, under **Properties**, for the corresponding fluid material of the continuous phase mixture). Note that for the definition of the evaporating species properties, the **user-defined** method is not supported and should not be used.

The default and recommended value for the averaging coefficient is 1/3. However, to improve the simulation results, you must provide accurate temperature-dependent data for the vapor material (through the **Create/Edit Materials** dialog box).

Cp

is the specific heat, c_p, of the particle. The specific heat may be defined as a function of temperature by selecting one of the function types from the drop-down list to the right of **Cp**. See Defining Properties Using Temperature-Dependent Functions (p. 475) for details about temperature-dependent properties. For multicomponent particles, it can be calculated as a mass-weighted value of the specific heat of the droplet component.

You also have the option of implementing a user-defined function to model the particle specific heat. See `DEFINE_DPM_PROPERTY` in the *Fluent Customization Manual* for more information.

A composition-dependent char specific heat option can be enabled if you are using the multiple-surface-reactions model for a combusting particle. For details on enabling this model, see **Combustion Model** (p. 1319).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you are using the non-premixed or the partially-premixed combustion model in the continuous phase calculation, the specific heat defined for the particle material will be used for the specific heat and enthalpy calculations of the non-volatile/non-reacting particle mass.

Density

is the density of the particulate phase in units of mass per unit volume of the discrete phase. This density is the mass density and not the volumetric density. The density may be defined as a function of temperature by selecting one of the function types from the drop-down list to the right of **Density**. See Defining Properties Using Temperature-Dependent Functions (p. 475) for details about temperature-dependent properties. For compressible flows, or if any of the real-gas models is enabled, the **compressible-liquid** method is also available. See Compressible Liquid Density Method (p. 480) for details. For multicomponent particles, it can be calculated as a volume-weighted value of the density of the droplet components.

You also have the option of implementing a user-defined function to model the particle density. See `DEFINE_DPM_PROPERTY` in the *Fluent Customization Manual* for more information.

For a combusting particle that swells during the trajectory calculations, the temperature-dependent density calculation is suspended during the devolatilization law and your input is used to determine the initial particle diameter, $d_{p,0}$ at the start of the devolatilization in Equation 16.157 in the *Fluent Theory Guide*. A composition-dependent char density option can be enabled if you are using the multiple-surface-reactions model for a combusting particle. For details on enabling this model, see **Combustion Model** (p. 1319).

Devolatilization Model

defines which version of the devolatilization model, Law 4, is being used. If you want to use the default constant rate devolatilization model, Equation 16.126 in the Theory Guide, retain the selection of **constant** in the drop-down list to the right of **Devolatilization Model** and input the rate constant A_0 in the field below the list.

You can choose one of the optional devolatilization models (the single kinetic rate, two kinetic rates, or CPD model, as described in Devolatilization (Law 4) in the Theory Guide) by selecting **single rate**, **two-competing-rates**, or **cpd-model** in the drop-down list.

When the single kinetic rate model (**single-rate**) is selected, the Single Rate Devolatilization Dialog Box (p. 2350) will appear and you will enter the **Pre-exponential Factor**, A_1, and the **Activation Energy**, E, to be used in Equation 16.128 in the Theory Guide for the computation of the kinetic rate.

When the two competing rates model (**two-competing-rates**) is selected, the Two Competing Rates Model Dialog Box (p. 2351) will appear and you will enter, for the **First Rate** and the **Second Rate**, the **Pre-exponential Factor** (A_1 in Equation 16.130 and A_2 in Equation 16.131 in the Theory Guide), **Activation Energy** (E_1 in Equation 16.130 and E_2 in Equation 16.131), and **Weighting Factor** (α_1 and α_2 in Equation 16.132). The constants you specify are used in Equation 16.130 through Equation 16.132.

When the CPD model (**cpd-model**) is selected, the **CPD Model** dialog box will appear and you will enter the **Initial Fraction of Bridges in Coal Lattice** (p_0 in Equation 16.143 of the Theory Guide), **Initial Fraction of Char Bridges** (c_0 in Equation 16.142), **Lattice Coordination Number** ($\sigma+1$ in Equation 16.154), **Cluster Molecular Weight** ($M_{w,1}$ in Equation 16.154), and **Side Chain Molecular Weight** ($M_{w,\delta}$ in Equation 16.153).

Note that the **Single Rate Devolatilization Model**, **Two Competing Rates Model**, and **CPD Model** dialog boxes are modal dialog boxes, which means that you must tend to them immediately before continuing the property definitions.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Diffusivity Reference Pressure

is the reference pressure P_{ref} for the pressure dependent binary diffusivity in Equation 16.110 in the *Fluent Theory Guide*. To apply the pressure dependence, in the **Create/Edit Materials** dialog box, under **Properties**, from the **Diffusivity Reference Pressure** drop-down list, select `constant` and enter a reference pressure value corresponding to the value for the **Binary Diffusivity** material property. If `none` is selected from the **Diffusivity Reference Pressure** drop-down list, ANSYS Fluent assumes no pressure dependence for the binary diffusivity.

Heat of Pyrolysis

is the heat of the instantaneous pyrolysis reaction , h_{pyrol}, that the evaporating/boiling species may undergo when released to the continuous phase. This input represents the conversion of the evaporating species to lighter components during the evaporation process. The heat of pyrolysis should be specified as a positive number for exothermic reaction and as a negative number for endothermic reaction. The default value of zero implies that the heat of pyrolysis is not considered. This input is used in Equation 16.436 in the Theory Guide.

Heat of Reaction for Burnout

is the heat released by the surface char combustion reaction, Law 5 (Equation 16.164 in the Theory Guide). This parameter is specified in terms of heat release (for example, Joules) per unit mass of char consumed in the surface reaction.

Latent Heat

is the latent heat of vaporization, h_{fg}, required for phase change from an evaporating liquid droplet (Equation 16.112, Equation 16.285, Equation 16.286, and Equation 16.300 in the Theory Guide) or for the evolution of volatiles from a combusting particle (Equation 16.158 in the Theory Guide). This input is supplied in units of J/kg in SI units or of Btu/lb_m in British units and is treated as a constant by ANSYS Fluent. For the droplet particle, the latent heat value at the boiling point temperature should be used.

You also have the option of implementing a user-defined function to model the particle latent heat. See `DEFINE_DPM_PROPERTY` in the *Fluent Customization Manual* for more information.

React. Heat Fraction Absorbed by Solid

is the parameter f_h (Equation 16.178 in the Theory Guide), which controls the distribution of the heat of reaction between the particle and the continuous phase. The default value of zero implies that the entire heat of reaction is released to the continuous phase.

Saturation Vapor Pressure

is the saturated vapor pressure, p_{sat}, defined as a function of temperature, which is used in the vaporization law, Law 2 (Equation 16.102 in the Theory Guide). The saturated vapor pressure may be defined as a function of temperature by selecting one of the function types from the drop-down list to the right of its name. (See Defining Properties Using Temperature-Dependent Functions (p. 475) for details about temperature-dependent properties.) In the case of unrealistic inputs, ANSYS Fluent restricts the range of P_{sat} to between 0.0 and the operating pressure. Correct specification of a realistic vapor pressure curve is essential for accurate results from the vaporization model.

You also have the option of implementing a user-defined function to model the particle saturation vapor pressure. See `DEFINE_DPM_PROPERTY` in the *Fluent Customization Manual* for more information.

Swelling Coefficient

is the coefficient C_{sw} in Equation 16.157 in the Theory Guide, which governs the swelling of the coal particle during the devolatilization law, Law 4 (Devolatilization (Law 4) in the Theory Guide). A swelling coefficient

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of unity (the default) implies that the coal particle stays at constant diameter during the devolatilization process.

You also have the option of implementing a user-defined function to model the particle swelling coefficient. See DEFINE_DPM_PROPERTY in the *Fluent Customization Manual* for more information.

Thermal Conductivity

is the thermal conductivity of the particle, k_p. This input is specified in units of W/m-K in SI units or $^{\text{Btu}}/_{\text{ft-h-}^0\text{F}}$ in British units and is treated as a constant by ANSYS Fluent.

Temperature Averaging Coefficient,

is the coefficient α_T in Equation 16.200 in the *Fluent Theory Guide*. To assume bulk gas temperature for the physical properties in the particle vaporization and heating rates equations, select none from the **Temperature Averaging Coefficient** drop-down in the **Create/Edit Materials** dialog box, under **Properties**. To enable property averaging, select constant and enter a value between 0 and 1 for the **Temperature Averaging Coefficient**. α_T= 1 corresponds to bulk gas conditions. For α_T= 0, the temperature reverts to the particle surface temperature.

Important

When you define the **Temperature Averaging Coefficient** with a constant value other than 1, you must also define **Specific Heat**, **Viscosity** and **Thermal Conductivity** for the evaporating species (in the **Create/Edit Materials** dialog box, under **Properties**, for the corresponding fluid material of the continuous phase mixture). Note that for the definition of the evaporating species properties, the **user-defined** method is not supported and should not be used.

The default and recommended value for the averaging coefficient is 1/3. However, to improve the simulation results, you must provide accurate temperature-dependent data for the evaporating vapor material (through the **Create/Edit Materials** dialog box).

Thermophoretic Coefficient

is the coefficient $D_{T,p}$ in Equation 16.11 in the Theory Guide, and appears when the thermophoretic force (which is described in Thermophoretic Force in the Theory Guide) is included in the trajectory calculation (that is, when the **Thermophoretic Force** option is enabled in the **Discrete Phase Model** dialog box). The default is the expression developed by Talbot [117] (p. 2893) (**talbot-diffusion-coeff**) and requires no input from you. You can also define the thermophoretic coefficient as a function of temperature by selecting one of the function types from the drop-down list to the right of **Thermophoretic Coefficient**. See Defining Properties Using Temperature-Dependent Functions (p. 475) for details about temperature-dependent properties.

You also have the option of implementing a user-defined function to model the particle thermophoretic coefficient. See DEFINE_DPM_PROPERTY in the *Fluent Customization Manual* for more information.

Vaporization Temperature

is the temperature, T_{vap}, at which the calculation of vaporization from a liquid droplet or devolatilization from a combusting particle is initiated by ANSYS Fluent. Until the particle temperature reaches T_{vap}, the particle is heated via Law 1, Equation 16.92 in the Theory Guide. This temperature input represents a modeling decision rather than any physical characteristic of the discrete phase.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You also have the option of implementing a user-defined function to model the particle vaporization temperature. See DEFINE_DPM_PROPERTY in the *Fluent Customization Manual* for more information.

Vapor-Particle-Equilibrium

is the selected approach for the calculation of the vapor concentration of the components at the surface. This can be Raoult's law (Equation 16.191 in the Theory Guide), the Peng-Robinson real gas model (Equation 16.199 in the Theory Guide), or a user-defined function that defines the equilibrium.

Vaporization Model

defines which vaporization model is used for pure droplets (Law 2) and for multicomponent droplets (Law 7). If you want to use the default diffusion controlled model, retain the selection of **diffusion-controlled** from the drop-down list to the right of **Vaporization Model**. This will apply Equation 16.101 in the Theory Guide.

To use the convection/diffusion controlled model for vaporization select **convection/diffusion-controlled** from the drop-down list. Equation 16.106 in the Theory Guide will be applied for the calculation of the vaporization rate, and Equation 16.116 in the Theory Guide will be applied in the particle heat transfer calculations This model is recommended when evaporation rates are high. For slowly evaporating droplets both models are expected to give similar results.

If you have selected the **convection/diffusion-controlled**, you can enable the **Variable Lewis Number Formulation** option in the **Convection/Diffusion Model** dialog box to compute the Spalding heat number from Equation 16.115 in the *Fluent Theory Guide*. If the option is disabled ANSYS Fluent uses Equation 16.116.

You can also enable averaging of the Spalding heat transfer term for the convection/diffusion-controlled model according to Equation 16.117 in the *Fluent Theory Guide* by using the TUI command `define/models/dpm/options/vaporization-heat-transfer-averaging`. This option has the net effect of increasing the heat-transfer to the droplet and may produce more realistic results for cases with large temperature differences between droplet and bulk gas.

Volatile Component Fraction

(f_{v0}) is the mass fraction of a droplet particle that may vaporize via Laws 2 and/or 3 (Droplet Vaporization (Law 2) in the Theory Guide). For combusting particles, it is the mass fraction of volatiles that may be evolved via Law 4 (Devolatilization (Law 4) in the Theory Guide).

When the effect of particles on radiation is enabled (for the P-1 or discrete ordinates radiation model only) in the Discrete Phase Model Dialog Box (p. 2278), you must define the following additional parameters:

Particle Emissivity

is the emissivity of particles in your model, ε_p, used to compute radiation heat transfer to the particles (Equation 16.92, Equation 16.112, Equation 16.123, Equation 16.158, and Equation 16.178 in the Theory Guide) when the P-1 or discrete ordinates radiation model is active. Note that you must enable radiation to particles, using the **Particle Radiation Interaction** option in the Discrete Phase Model Dialog Box (p. 2278). Recommended values of particle emissivity are 1.0 for coal particles and 0.5 for ash [62] (p. 2890).

You also have the option of implementing a user-defined function to model the particle emissivity. See DEFINE_DPM_PROPERTY in the *Fluent Customization Manual* for more information.

Particle Scattering Factor

is the scattering factor, f_p, due to particles in the P-1 or discrete ordinates radiation model (Equation 5.34 in the Theory Guide). Note that you must enable particle effects in the radiation model, using the **Particle Radiation Interaction** option in the Discrete Phase Model Dialog Box (p. 2278). The recommended value of

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

f_p for coal combustion modeling is 0.9 [62] (p. 2890). Note that if the effect of particles on radiation is enabled, scattering in the continuous phase will be ignored in the radiation model.

You also have the option of implementing a user-defined function to model the particle scattering factor. See `DEFINE_DPM_PROPERTY` in the *Fluent Customization Manual* for more information.

When an atomizer injection model and/or the droplet breakup or collision model is enabled in the Set Injection Properties Dialog Box (p. 2796) (atomizers) and/or Discrete Phase Model Dialog Box (p. 2278) (droplet breakup/collision), you must define the following additional parameters:

Droplet Surface Tension

is the droplet surface tension, σ. The surface tension may be defined as a function of temperature by selecting one of the function types from the drop-down list to the right of **Droplet Surface Tension**. See Defining Properties Using Temperature-Dependent Functions (p. 475) for details about temperature-dependent properties. You also have the option of implementing a user-defined function to model the droplet surface tension. More information about user-defined functions can be found in the Fluent Customization Manual.

Viscosity

is the droplet viscosity, μ_l. The viscosity may be defined as a function of temperature by selecting one of the function types from the drop-down list to the right of **Viscosity**. See Defining Properties Using Temperature-Dependent Functions (p. 475) for details about temperature-dependent properties. You also have the option of implementing a user-defined function to model the droplet viscosity. More information about user-defined functions can be found in the Fluent Customization Manual.

23.6. Solution Strategies for the Discrete Phase

Solution of the discrete phase implies integration in time of the force balance on the particle (Equation 16.1 in the Theory Guide) to yield the particle trajectory. As the particle is moved along its trajectory, heat and mass transfer between the particle and the continuous phase are also computed via the heat/mass transfer laws (Laws for Heat and Mass Exchange in the Theory Guide). The accuracy of the discrete phase calculation therefore depends on the time accuracy of the integration and upon the appropriate coupling between the discrete and continuous phases when required. Numerical controls are described in Numerics of the Discrete Phase Model (p. 1260). Coupling and performing trajectory calculations are described in Performing Trajectory Calculations (p. 1325). Resetting the Interphase Exchange Terms (p. 1329) and Parallel Processing for the Discrete Phase Model (p. 1360) provide information about resetting interphase exchange terms and using the parallel solver for a discrete phase calculation.

For additional information, see the following sections:

23.6.1. Performing Trajectory Calculations
23.6.2. Resetting the Interphase Exchange Terms

23.6.1. Performing Trajectory Calculations

The trajectories of your discrete phase injections are computed when you display the trajectories using graphics or when you perform solution iterations. That is, you can display trajectories without impacting the continuous phase, or you can include their effect on the continuum (termed a coupled calculation). In turbulent flows, trajectories can be based on mean (time-averaged) continuous phase velocities or they can be impacted by instantaneous velocity fluctuations in the fluid. This section describes the procedures and commands you use to perform coupled or uncoupled trajectory calculations, with or without stochastic tracking or cloud tracking.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23.6.1.1. Uncoupled Calculations

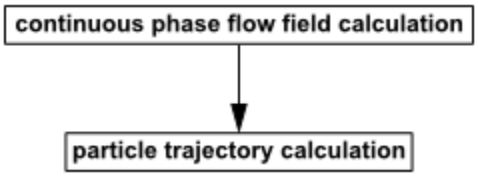
For the uncoupled calculation, you will perform the following two steps:

1. Solve the continuous phase flow field.

2. Plot (and report) the particle trajectories for discrete phase injections of interest.

In the uncoupled approach, this two-step procedure completes the modeling effort, as illustrated in Figure 23.34: Uncoupled Discrete Phase Calculations (p. 1326). The particle trajectories are computed as they are displayed, based on a fixed continuous-phase flow field. Graphical and reporting options are detailed in Postprocessing for the Discrete Phase (p. 1329).

Figure 23.34: Uncoupled Discrete Phase Calculations



This procedure is adequate when the discrete phase is present at a low mass and momentum loading, in which case the continuous phase is not impacted by the presence of the discrete phase.

23.6.1.2. Coupled Calculations

In a coupled two-phase simulation, ANSYS Fluent modifies the two-step procedure above as follows:

1. Solve the continuous phase flow field (prior to introduction of the discrete phase).

2. Introduce the discrete phase by calculating the particle trajectories for each discrete phase injection.

3. Recalculate the continuous phase flow, using the interphase exchange of momentum, heat, and mass determined during the previous particle calculation.

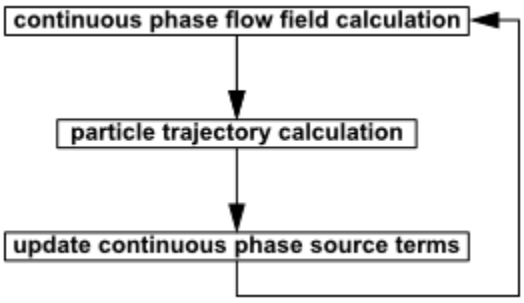
4. Recalculate the discrete phase trajectories in the modified continuous phase flow field.

5. Repeat the previous two steps until a converged solution is achieved in which both the continuous phase flow field and the discrete phase particle trajectories are unchanged with each additional calculation.

This coupled calculation procedure is illustrated in Figure 23.35: Coupled Discrete Phase Calculations (p. 1327). When your ANSYS Fluent model includes a high mass and/or momentum loading in the discrete phase, the coupled procedure must be followed in order to include the important impact of the discrete phase on the continuous phase flow field.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.35: Coupled Discrete Phase Calculations



Important

When you perform coupled calculations, all defined discrete phase injections will be computed. You cannot calculate a subset of the injections you have defined. If there are massless particle injections defined, these will have no effect in the coupled calculation.

23.6.1.2.1. Procedures for a Coupled Two-Phase Flow

If your ANSYS Fluent model includes prediction of a coupled two-phase flow, you should begin with a partially (or fully) converged continuous-phase flow field. You will then create your injection(s) and set up the coupled calculation.

For each discrete-phase iteration, ANSYS Fluent computes the particle/droplet trajectories and updates the interphase exchange of momentum, heat, and mass in each control volume. These interphase exchange terms then impact the continuous phase when the continuous phase iteration is performed. During the coupled calculation, ANSYS Fluent will perform the discrete phase iteration at specified intervals during the continuous-phase calculation. The coupled calculation continues until the continuous phase flow field no longer changes with further calculations (that is, all convergence criteria are satisfied). When convergence is reached, the discrete phase trajectories no longer change either, since changes in the discrete phase trajectories would result in changes in the continuous phase flow field.

The steps for setting up the coupled calculation are as follows:

1. Solve the continuous phase flow field.

2. In the Discrete Phase Model Dialog Box (p. 2278) (Figure 23.1: The Discrete Phase Model Dialog Box and the Tracking Parameters (p. 1247)), enable the **Interaction with Continuous Phase** option.

3. Set the frequency with which the particle trajectory calculations are introduced in the **DPM Iteration Interval** field. If you set this parameter to 5, for example, a discrete phase iteration will be performed every fifth continuous phase iteration. The optimum number of iterations between trajectory calculations depends upon the physics of your ANSYS Fluent model.

   Important

   Note that if you set this parameter to 0, ANSYS Fluent will not perform any discrete phase iterations.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

During the coupled calculation (which you initiate using the Run Calculation Task Page (p. 2537) in the usual manner) you will see the following information in the ANSYS Fluent console as the continuous and discrete phase iterations are performed:

```
 iter   continuity   x-velocity   y-velocity       k         epsilon       energy    time/it
  314   2.5249e-01   2.8657e-01   1.0533e+00   7.6227e-02   2.9771e-02   9.8181e-03   :00:05
  315   2.7955e-01   2.5867e-01   9.2736e-01   6.4516e-02   2.6545e-02   4.2314e-03   :00:03

    DPM  Iteration  ....
    number  tracked=  9,  number  escaped=  1,  aborted=  0,  trapped=  0,  evaporated  =  8,i
 Done.
  316   1.9206e-01   1.1860e-01   6.9573e-01   5.2692e-02   2.3997e-02   2.4532e-03   :00:02
  317   2.0729e-01   3.2982e-02   8.3036e-01   4.1649e-02   2.2111e-02   2.5369e-01   :00:01
  318   3.2820e-01   5.5508e-02   6.0900e-01   5.9018e-02   2.6619e-02   4.0394e-02   :00:00
```

Note that you can perform a discrete phase calculation at any time by using the `solve/dpm-update` text command.

23.6.1.2.2. Stochastic Tracking in Coupled Calculations

If you include the stochastic prediction of turbulent dispersion in the coupled two-phase flow calculations, the number of stochastic tries applied each time the discrete phase trajectories are introduced during coupled calculations will be equal to the **Number of Tries** specified in the Set Injection Properties Dialog Box (p. 2796). For transient particle tracking, the number of tries is set to 1. Input of this parameter is described in Stochastic Tracking (p. 1294). An input of $n \geq 1$ requests n stochastic trajectory calculations for each particle in the injection. When the number of stochastic tracks included is small, you may find that the ensemble average of the trajectories is quite different each time the trajectories are computed. These differences may, in turn, impact the convergence of your coupled solution. For good convergence of a coupled solution, a statistical independent distribution of tracks can be achieved with an adequate number of stochastic tracks and/or a sufficient number of different starting locations of the tracks.

23.6.1.2.3. Under-Relaxation of the Interphase Exchange Terms

When you are coupling the discrete and continuous phases using the calculation procedures noted above, ANSYS Fluent applies under-relaxation to the momentum, heat, and mass transfer terms. This under-relaxation serves to increase the stability of the coupled calculation procedure by letting the impact of the discrete phase change only gradually:

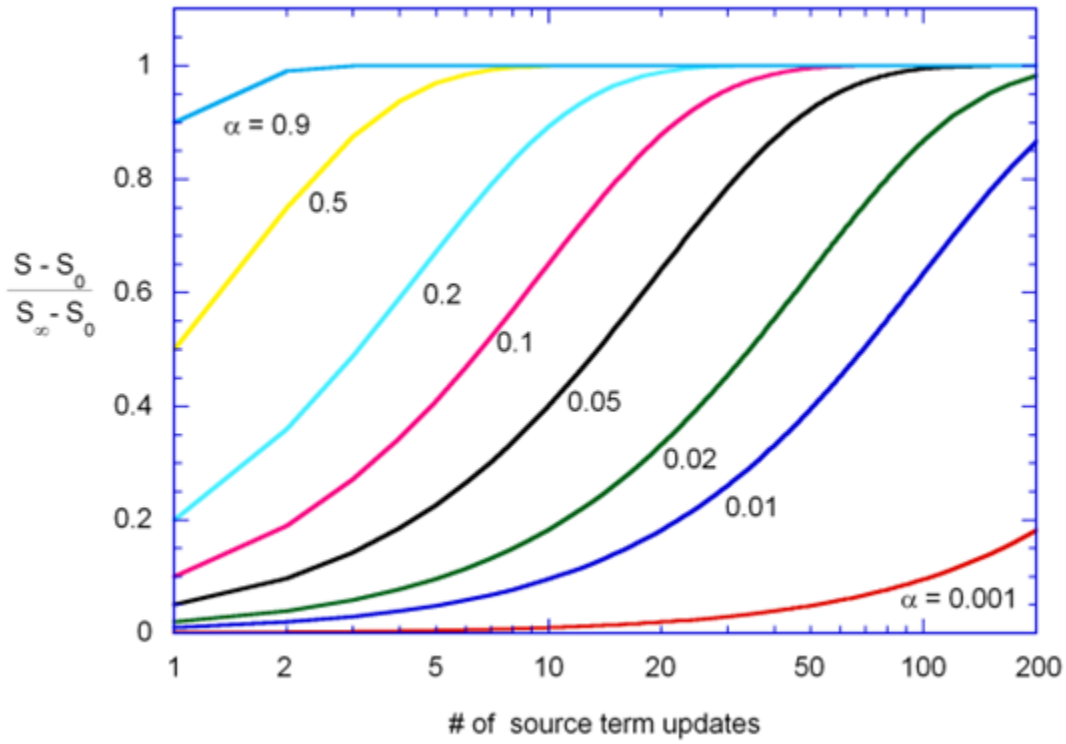
$$E_{new} = E_{old} + \alpha \left(E_{calculated} - E_{old} \right) \tag{23.13}$$

where E_{new} is the exchange term, E_{old} is the previous value, $E_{calculated}$ is the newly computed value, and α is the particle/droplet under-relaxation factor. For α, ANSYS Fluent uses a default value of 0.5 for all types of analysis except transient flows with unsteady particle tracking, in which case the default value of 0.9 is used. You can modify α by changing the value in the **Discrete Phase Sources** field under **Under-Relaxation Factors** in the **Solution Controls** task page. You may need to decrease α in order to improve the stability of coupled discrete phase calculations.

Figure 23.36: Effect of Number of Source Term Updates on Source Term Applied to Flow Equations (p. 1329) shows how the source term, S, when applied to the flow equations, changes with the number of updates for varying under-relaxation factors. In Figure 23.36: Effect of Number of Source Term Updates on Source Term Applied to Flow Equations (p. 1329), S_∞ is the final source term for which a value is reached after a certain number of updates and S_0 is the initial source term at the start of the computation. The value of S_0 is typically zero at the beginning of the calculation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.36: Effect of Number of Source Term Updates on Source Term Applied to Flow Equations



In a continuous flow simulation, with continuous DPM tracking, suppose that the DPM under-relaxation factor is chosen to be 0.5, with 20 continuous phase iterations per DPM iteration. From Figure 23.36: Effect of Number of Source Term Updates on Source Term Applied to Flow Equations (p. 1329), we see that approximately 10 source term updates are required for the DPM sources to reach their final values. Therefore, in this example, at least 200 continuous phase iterations are required after any change to the DPM sources (for example, a new injection or a changed DPM mass flow rate), to ensure that the change has taken effect.

23.6.2. Resetting the Interphase Exchange Terms

If you have performed coupled calculations, resulting in nonzero interphase sources/sinks of momentum, heat, and/or mass that you do not want to include in subsequent calculations, you can reset these sources to zero.

Solution → Initialization → Reset DPM Sources

When you select **Reset DPM Sources**, the sources will immediately be reset to zero without any further confirmation from you.

23.7. Postprocessing for the Discrete Phase

After you have completed your discrete phase inputs and any coupled two-phase calculations of interest, you can display and store the particle trajectory predictions. ANSYS Fluent provides both graphical and alphanumeric reporting facilities for the discrete phase, including the following:

- graphical display of the particle trajectories

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- summary reports of trajectory fates
- step-by-step reports of the particle position, velocity, temperature, and diameter
- alphanumeric reports and graphical display of the interphase exchange of momentum, heat, and mass
- optionally, alphanumeric reports and graphical display of various cell-averaged discrete phase field variables
- sampling of trajectories at boundaries and lines/planes
- summary reporting of current particles in the domain
- histograms of trajectory data at sample planes
- display of erosion/accretion rates
- exporting of trajectories to Fieldview and Ensight

This section provides detailed descriptions of each of these postprocessing options.

(Note that plotting or reporting trajectories does not change the source terms.)

For additional information, see the following sections:
23.7.1. Displaying of Trajectories
23.7.2. Reporting of Trajectory Fates
23.7.3. Step-by-Step Reporting of Trajectories
23.7.4. Reporting of Current Positions for Unsteady Tracking
23.7.5. Reporting of Interphase Exchange Terms (Discrete Phase Sources)
23.7.6. Reporting of Discrete Phase Variables
23.7.7. Reporting of Unsteady DPM Statistics
23.7.8. Sampling of Trajectories
23.7.9. Histogram Reporting of Samples
23.7.10. Summary Reporting of Current Particles
23.7.11. Postprocessing of Erosion/Accretion Rates

23.7.1. Displaying of Trajectories

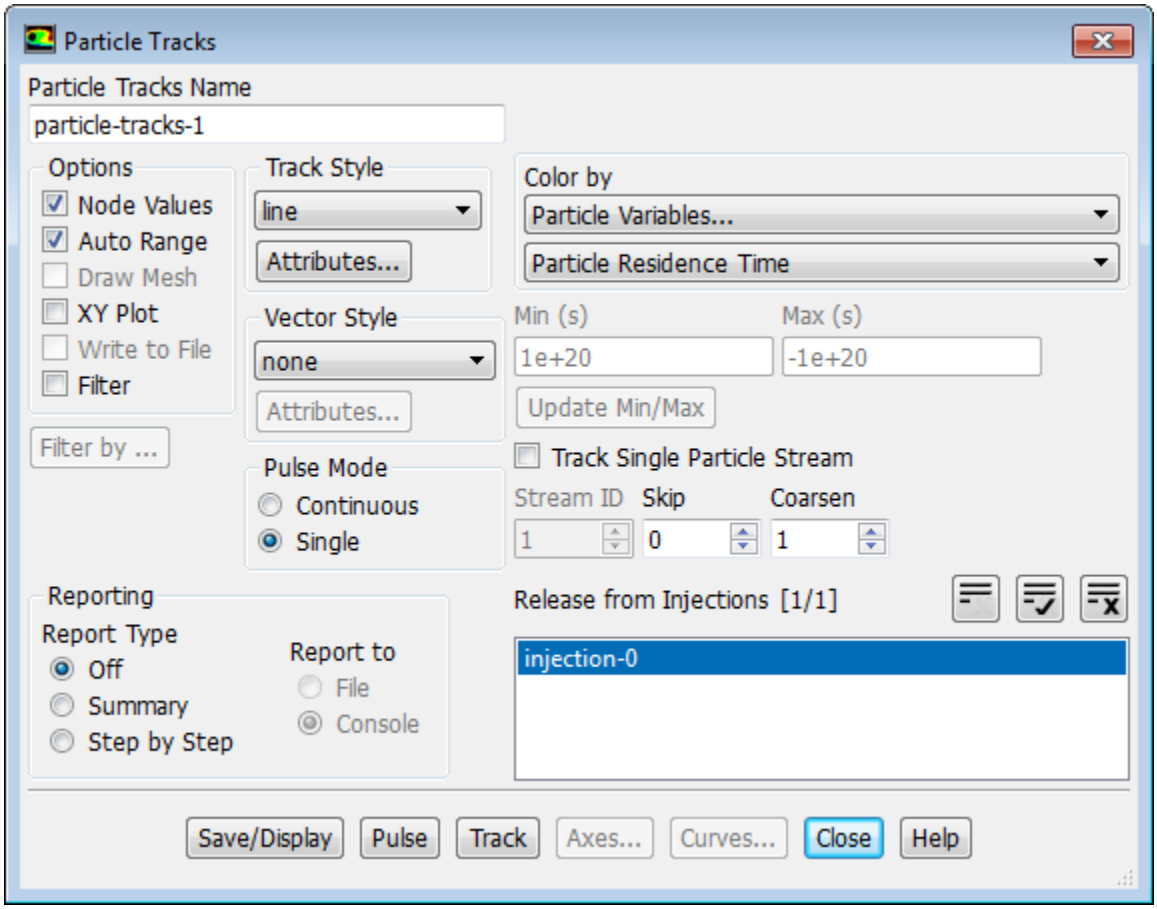
When you have defined discrete phase particle injections, as described in Setting Initial Conditions for the Discrete Phase (p. 1265), you can display the trajectories of these discrete particles using the Particle Tracks Dialog Box (p. 2761) (Figure 23.37: The Particle Tracks Dialog Box (p. 1331)).

You can create named particle track definitions and save them for later use. See Creating and Using Contour Plot Definitions (p. 1762) for additional information on graphics object definitions.

Results → Graphics → Particle Tracks → New...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.37: The Particle Tracks Dialog Box



The procedure for drawing trajectories for particle injections is as follows:

1. Select the particle injection(s) you want to track in the **Release from Injections** list. (You can choose to track a specific particle, instead, as described below.)

2. Set the length scale and the maximum number of steps in the Discrete Phase Model Dialog Box (p. 2278), as described in Numerics of the Discrete Phase Model (p. 1260).

   Setup → Models → Discrete Phase Edit...

   If stochastic and/or cloud tracking is desired, set the related parameters in the **Set Injection Properties** dialog box, as described in Stochastic Tracking (p. 1294).

   Note

   Displaying tracks multiple times at the same solution state will result in identical random tracks. This is because the same random seeds are used enabling you to postprocess different variables on the same particle tracks.

3. Set any of the display options described below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click the **Display** button to draw the trajectories or click the **Pulse** button to animate the particle positions. The **Pulse** button will become the **Stop !** button during the animation, and you must click **Stop !** to stop the pulsing.

Important

For unsteady particle tracking simulations, clicking **Display** will show only the current location of the particles. Typically, you should select **point** in the **Track Style** drop-down list when displaying transient particle locations since individual positions will be displayed. The **Pulse** button option is not available for unsteady tracking.

23.7.1.1. Specifying Particles for Display

You can display the trajectory for an individual particle stream instead of for all the streams in a given injection. Additionally, you can visualize the trajectories for the wall film particles (if they are present in the simulation) and/or free stream particles. To do so, you must first determine which particle is of interest. Use the Injections Dialog Box (p. 2716) to list the particle streams in the desired injection, as described in Creating and Modifying Injections (p. 1284).

Setup → Models → Discrete Phase → Injections Edit...

Note the ID numbers listed in the first column of the listing printed in the ANSYS Fluent console. Then perform the following steps after step 1 above. In the Particle Tracks Dialog Box (p. 2761):

1. Enable **Free Stream Particles** if you want to display those types of particles. Note that this option is available only when the Lagrangian wall film model is enabled in the wall boundary conditions dialog box, under the **DPM** tab.

2. Enable **Wall Film Particles** if you want to display those types of particles. Note that this option is available only when the Lagrangian wall film model is enabled in the wall boundary conditions dialog box, under the **DPM** tab.

3. Enable the **Track Single Particle Stream** to display those types of particles.

4. In the **Stream ID** field, specify the ID number of the particle stream for which you want to plot the trajectory.

23.7.1.1.1. Controlling the Particle Tracking Style

Particle tracking can be displayed as lines (with or without arrows), ribbons, cylinders (coarse, medium, or fine), triangles, spheres, or a set of points. You can choose **line**, **line-arrows**, **point**, **sphere**, **ribbon**, **triangle**, **coarse-cylinder**, **medium-cylinder**, or **fine-cylinder** in the **Track Style** drop-down list in the **Particle Tracks** dialog box. Pulsing can be done only on **point**, **sphere**, or **line** styles.

Once you have selected the track style, click the **Attributes...** button to specify how you would like to display the particle tracks.

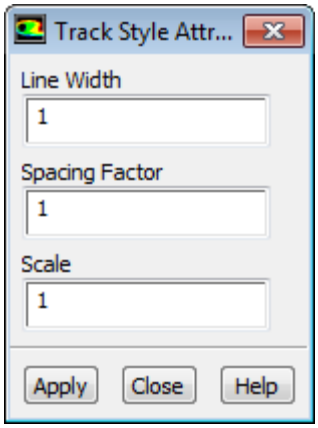
Note

The **Track Style** options that will appear depend on whether you are running a transient or steady-state simulation. For a transient case, the only **Track Style** options available are the **point** and **sphere** styles.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you are using the **line** or **line-arrows** style, set the **Line Width** in the **Track Style Attributes** dialog box (Figure 23.38: The Track Style Attributes Dialog Box (p. 1333)) that appears when you click the **Attributes...** button. For **line-arrows** you will also set the **Spacing Factor**, which controls the spacing between the particles tracks. The size of the arrow heads can be adjusted by entering a value in the **Scale** text-entry box.

  Figure 23.38: The Track Style Attributes Dialog Box



- If you are using the **point** style, you will set the **Marker Size** in the **Track Style Attributes** dialog box. The thickness of the particle track will be the thickness of the marker.

- If you are using the **sphere** style, you will set the **Diameter**, **scale**, and **Detail** in the **Particle Sphere Style Attributes** dialog box (Figure 23.39: The Particle Sphere Style Attributes Dialog Box (p. 1334)). You have the option of specifying a constant diameter if you enable **Constant** under **Options** and you will then specify the **Diameter**. If you enable **Variable**, you can select a particle variable to estimate the size of the spheres. The spheres are scaled by the factor entered in the **Scale** entry box.

  The best constant diameter to use will depend on the dimensions of the domain, the view, and the particle density. However, an adequate starting point would be a diameter on the order of 1/4 of the average cell size or 1/4 step size. Units for the **Diameter** field correspond to the mesh dimensional units.

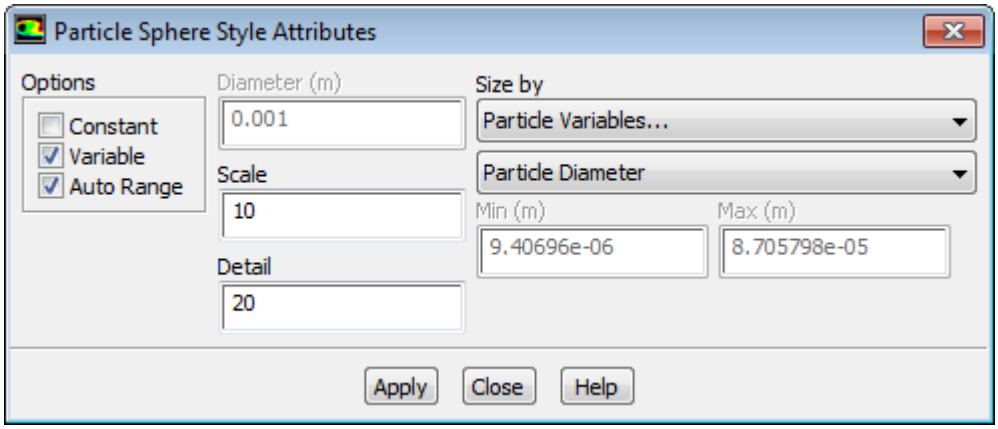
  The level of detail applied to the graphical rendering of the spheres can be controlled using the **Detail** field. The level of detail uses integer values ranging from 4 to 50. Note that the performance of the graphical rendering as well as the memory consumption will be better when using a small level of detail, that is, very coarse spheres, such as 6 or 8. The rendering performance significantly decreases with higher levels of detail. You should gradually increase the detail to determine the best-case scenario between performance and quality.

  Whenever **Auto Range** is disabled, the spheres are displayed only if they have values between **Min** and **Max**.

  Also note that to take full advantage of spherical rendering, lighting should be turned on in the view. The Gouraud setting provides much smoother looking spheres than the Flat setting and better performance than the Phong setting. For more information on lighting, see Adding Lights (p. 1796).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.39: The Particle Sphere Style Attributes Dialog Box



- If you are using the **triangle** or any of the **cylinder** styles, you will set the **Width** in the **Track Style Attributes** dialog box. For triangles, the specified value will be half the width of the triangle's base, and for cylinders, the value will be the cylinder's radius.

- If you are using the **ribbon** style, clicking on the **Attributes...** button will open the Ribbon Attributes Dialog Box (p. 2560), in which you can set the ribbon's **Width**. You can also specify parameters for twisting the ribbon tracks. In the **Twist By** drop-down list, you can select a scalar field on which the tracks twisting is based (for example, helicity). Select the desired category in the upper list and then select a related quantity in the lower list. The twisting may not be displayed smoothly because the scalar field by which you are twisting the tracks is calculated at cell centers only (and not interpolated to a particle's position). The **Twist Scale** sets the amount of twist for the selected scalar field. To magnify the twist for a field with very little change, increase this factor; to display less twist for a field with dramatic changes, decrease this factor.

  When you click **Compute**, the **Min** and **Max** fields will be updated to show the range of the **Twist By** scalar field.

23.7.1.1.2. Controlling the Vector Style of Particle Tracks

You can choose to have the particle tracks displayed as vectors. Choose the **Vector Style** from the drop-down list in the **Particle Tracks** dialog box:

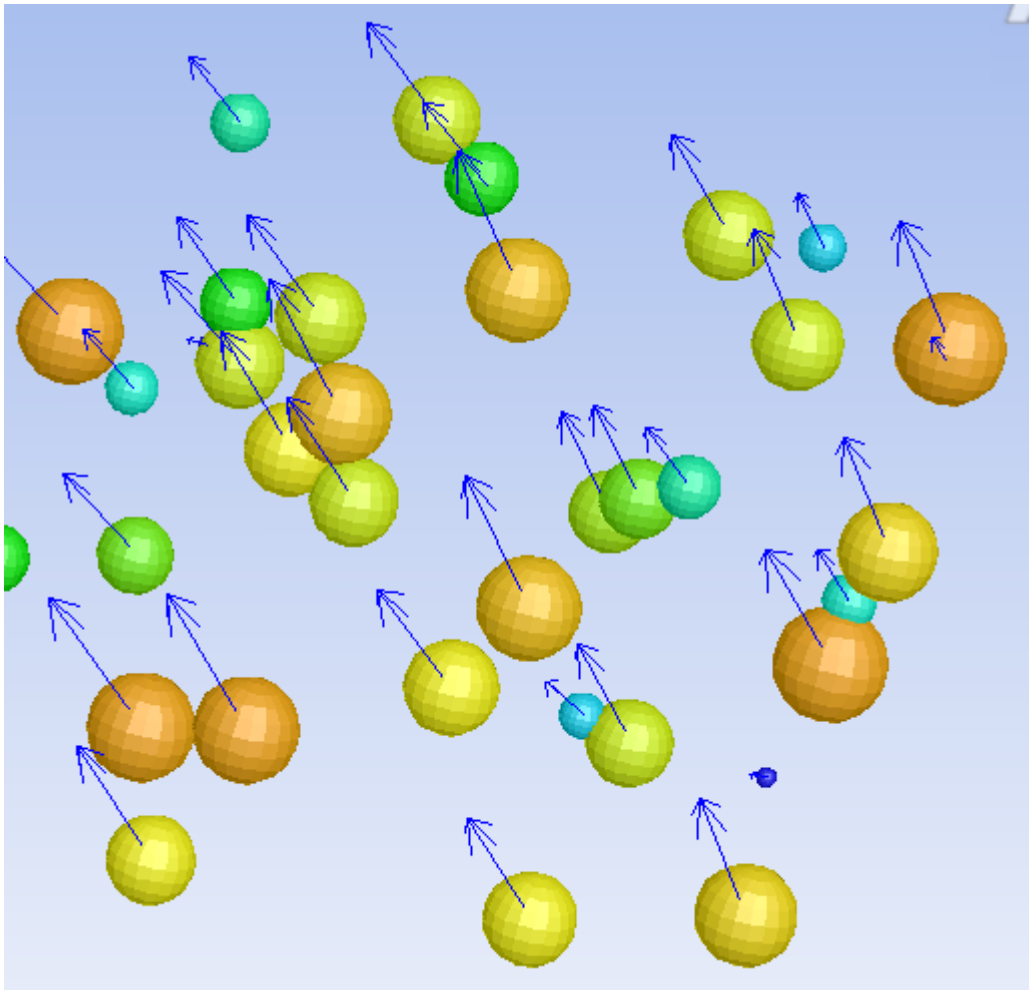
- If you select **vector**, the vector will be generated starting in the center of the particle, as shown in Figure 23.40: Particles with the Vector Style (p. 1335).

- If you select **centered-vector**, the midpoint of the vector will appear in the center of the particle, as shown in Figure 23.41: Particles with the Centered Vector Style (p. 1336).

- If you select **centered-cylinder**, the midpoint of the cylinder will appear in the center of the particle, as shown in Figure 23.42: Particles with the Centered Cylinder Style (p. 1337).

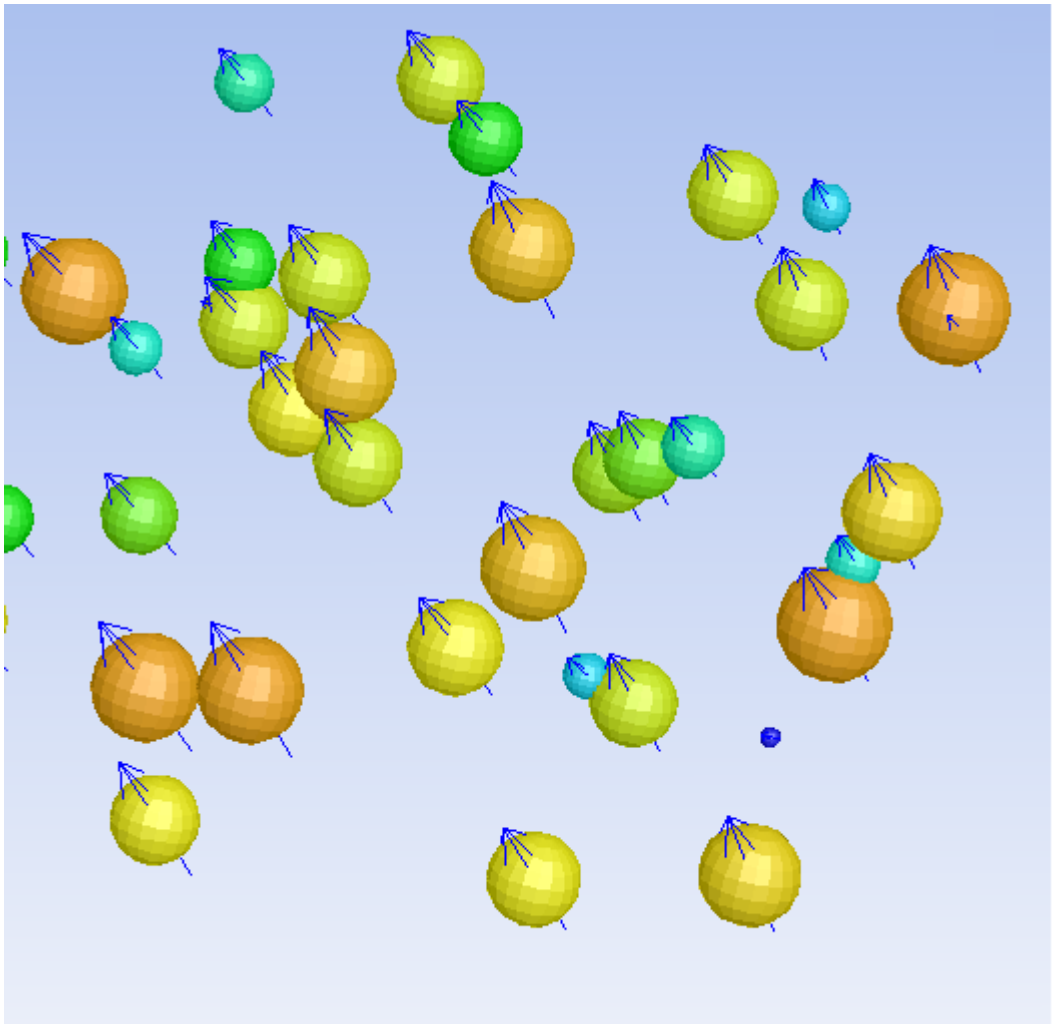
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.40: Particles with the Vector Style



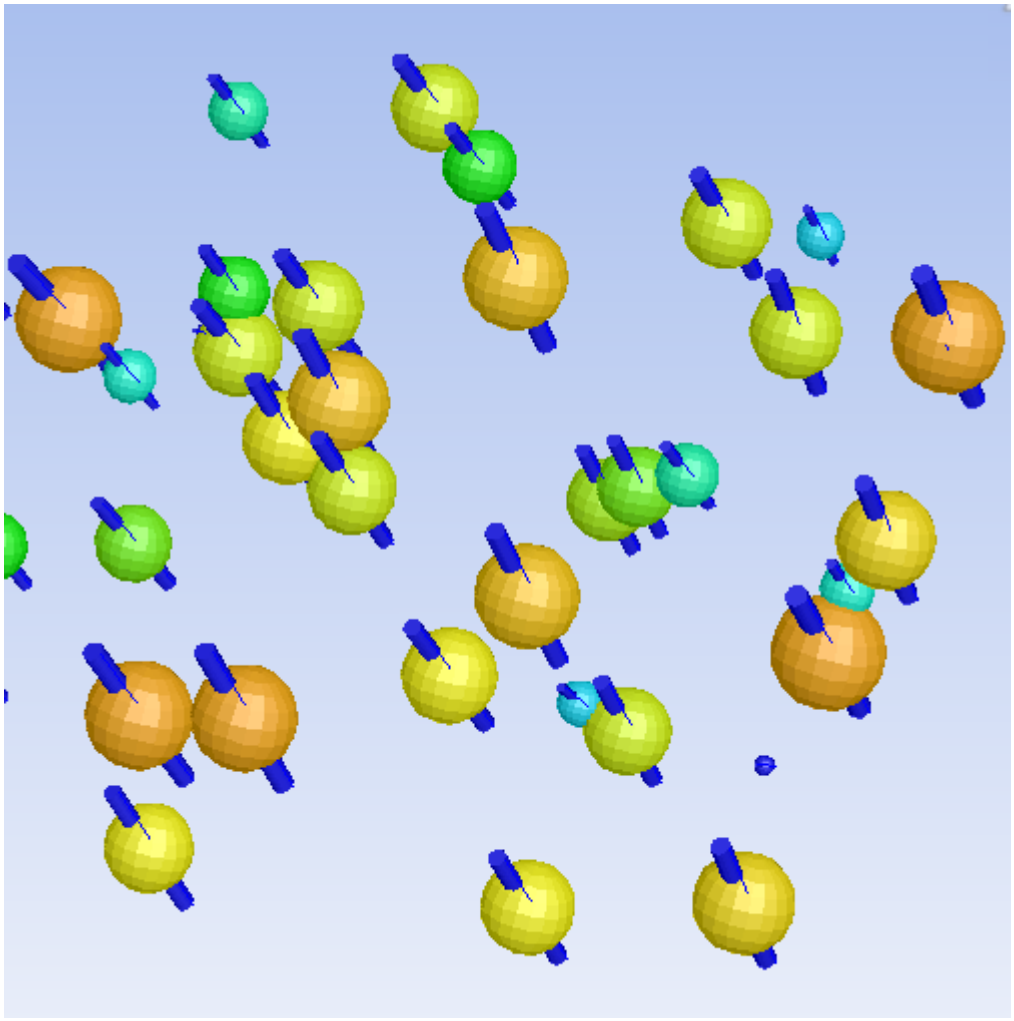
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.41: Particles with the Centered Vector Style



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.42: Particles with the Centered Cylinder Style

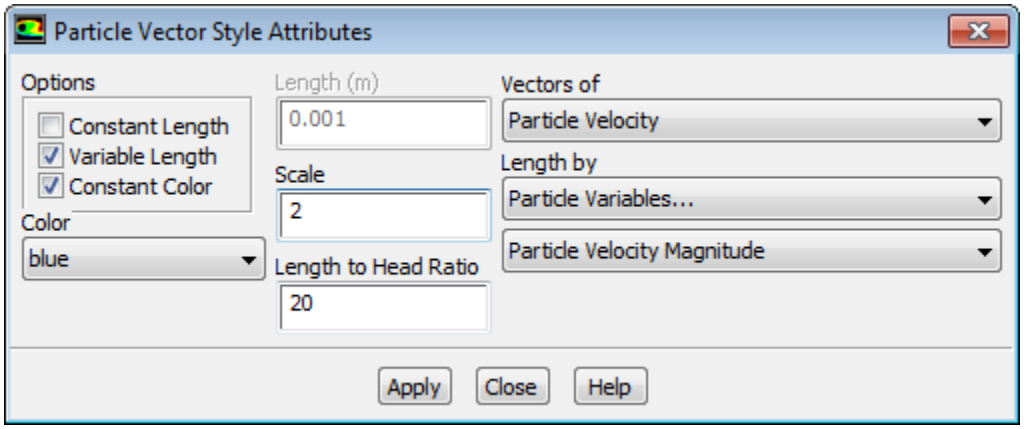


Click the **Attributes...** button to specify how you would like to display the particle tracks. In the **Particle Vector Style Attributes** dialog box (Figure 23.43: The Particle Vector Style Attributes Dialog Box (p. 1338)) you will set the **Length**, **Scale**, and **Length to Head Ratio**. The direction of the vectors is displayed for the selected variable under **Vectors of**. You have the option of specifying a **Constant Length** or a **Variable Length**, which is based on the variable selected under **Length by**. If **Constant Color** is enabled, then all vectors/cylinders are colored by the color selected in the **Color** drop-down list. Otherwise, it is the color selected in the **Particle Tracks** dialog box (seen in the **Mesh Colors** dialog box when **Draw Mesh** is enabled).

Vectors can be scaled by the factor given in the **Scale** entry box. The ratio of vector length to vector head size can be changed in the box **Length to Head Ratio**. In the case of a cylinder, the ratio of the length to the diameter is affected.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.43: The Particle Vector Style Attributes Dialog Box

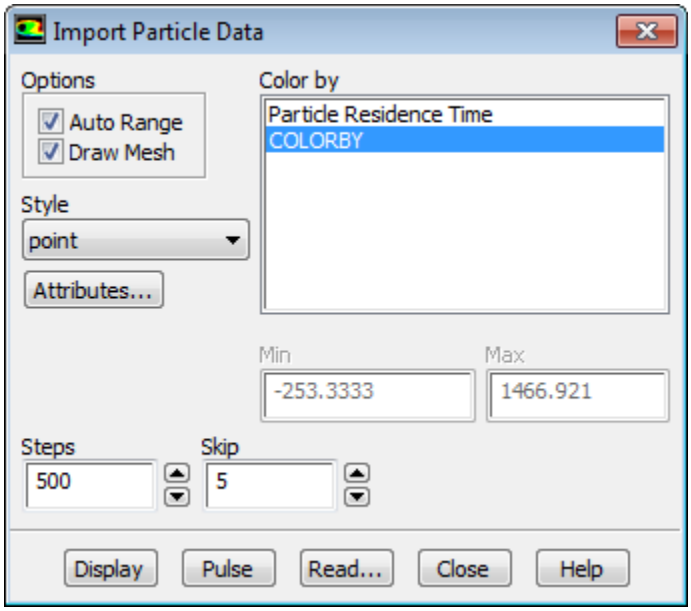


23.7.1.2. Importing Particle Data

Use the **Import Particle Data** dialog box (Figure 23.44: The Import Particle Data Dialog Box (p. 1338)) to import particle data to display in the graphics window.

Postprocessing → Model Specific → Discrete Phase → Import Particle Data...

Figure 23.44: The Import Particle Data Dialog Box



1. Click **Read...** to display a file selection dialog box where you can enter a file name and a directory that contains the imported data.

2. Choose from the available import options by selecting **Auto Range** and/or **Draw Mesh** under **Options**. If you prefer to restrict the range of the scalar field, disable the **Auto Range** option and set the **Min** and **Max** values manually beneath the **Color by** list.

3. Choose to color the particle pathlines by any of the scalar fields in the **Color by** list. If you select **COLORBY**, the pathlines will be colored by the quantity that was chosen when the particle data file was created. (See Exporting Steady-State Particle History Data (p. 84))

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Select a pathline style under **Style**. To set pathline style attributes, click the **Attributes...** button. For more information about the pathline style types, see Controlling the Pathline Style (p. 1774).

5. The value of **Steps** sets the maximum number of steps a particle can advance. A particle will stop when it has traveled this number of steps or when it leaves the domain.

6. If your pathline plot is difficult to understand because there are too many paths displayed, you can "thin out" the pathlines by changing the **Skip** value.

7. Click the **Display** button to draw the pathlines, or click the **Pulse** button to animate the particle positions. The **Pulse** button will become the **Stop !** button during the animation, and you must click **Stop !** to stop the pulsing.

23.7.1.3. Options for Particle Trajectory Plots

You can include the mesh in the trajectory display, control the style of the trajectories (including the twisting of ribbon-style trajectories), color them by different scalar fields and control the color scale, and coarsen trajectory plots. You can also choose node or cell values for display. If you are "pulsing" the trajectories, you can control the pulse mode. Finally, you can generate an XY plot of the particle trajectory data (for example, residence time) as a function of time or path length and save this XY plot data to a file.

Note

The **Draw Mesh** option settings are not saved with the particle track plot definition. Once the dialog box is closed these settings will revert to being disabled. If you want these settings persisted within the current session, you can use the non-persistent **Particle Tracks** dialog box.

Postprocessing → **Graphics** → **Particle Tracks** → **Edit...**

Plotting particle trajectories can be very time consuming, therefore, to reduce the plotting time, a coarsening factor can be used to reduce the number of points that are plotted. Providing a coarsening factor of n, will result in each n^{th} point being plotted for a given trajectory in any cell. This coarsening factor is specified in the Particle Tracks Dialog Box (p. 2761), in the **Coarsen** field and is only valid for steady-state cases. For example, if the coarsening factor is set to 2, then ANSYS Fluent will plot alternate points.

Important

Note that if any particle or pathline enters a new cell, this point will always be plotted.

To reduce plotting time in transient cases, ANSYS Fluent has available an option to skip plotting every n^{th} particle in an injection. Selecting this option is also done in the Particle Tracks Dialog Box (p. 2761) by specifying a nonzero integer in the **Skip** field. For example, if an individual stream is selected and the skip option is set to 1, every other particle will be plotted. If the entire injection is selected with a skip option of 1, every other particle will be plotted for all streams in the injection.

These options are controlled in exactly the same way that pathline-plotting options are controlled. See Options for Pathline Plots (p. 1773) for details about setting the trajectory plotting options mentioned above.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that in addition to coloring the trajectories by continuous phase variables, you can also color them according to the following discrete phase variables: particle time, particle velocity, particle diameter, particle density, particle mass, particle temperature, particle law number, particle time step, and particle Reynolds number. If **DEM Collisions** is enabled in the **Discrete Phase Model** dialog box, you can also select the magnitude or components of the collisional force acting on the DEM parcel, total force acting on the DEM parcel, and total acceleration experienced by the particles in the parcel. These variables are included in the **Particle Variables...** category of the **Color by** list. To display the minimum and maximum values in the domain, click the **Update Min/Max** button.

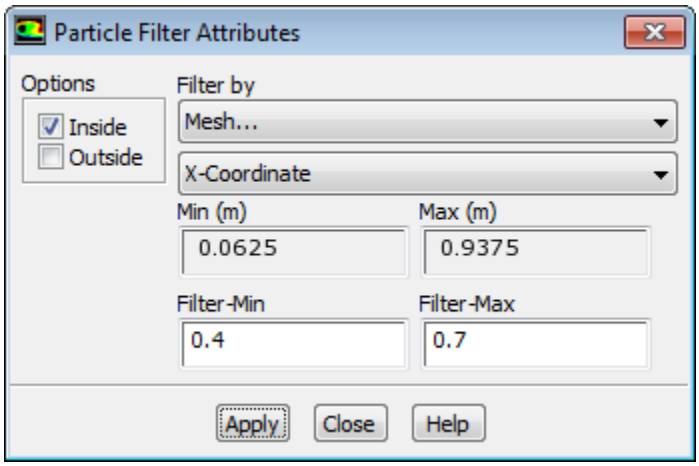
23.7.1.4. Particle Filtering

You can specify how you would like to filter the particles being displayed, by first enabling the **Enable Filter** option, then clicking the **Filter by...** button. In the **Particle Filter Attributes** dialog box, select the field variable by which you want to filter, then specify whether you would like to display all the particle tracks **Inside** or **Outside** the **Filter-Min** and **Filter-Max** range, as shown in Figure 23.45: The Particle Filter Attributes Dialog Box (p. 1340).

Note

All particle variables as well as any field variable except for **Custom Field Functions...** can be used as a filter variable.

Figure 23.45: The Particle Filter Attributes Dialog Box



23.7.1.5. Graphical Display for Axisymmetric Geometries

For axisymmetric problems in which the particle has a nonzero circumferential velocity component, the trajectory of an individual particle is often a spiral about the centerline of rotation. ANSYS Fluent displays the r and x components of the trajectory (but not the θ component) projected in the axisymmetric plane.

23.7.2. Reporting of Trajectory Fates

When you perform trajectory calculations by displaying the trajectories (as described in Displaying of Trajectories (p. 1330)), ANSYS Fluent will provide information about the trajectories as they are completed. By default, the number of trajectories with each possible fate (escaped, aborted, evaporated, and so on) is reported:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
DPM Iteration ....
 num. tracked = 7, escaped = 4, aborted = 0, trapped = 0, evaporated = 3, inco
Done.
```

You can also track particles through the domain without displaying the trajectories by clicking the **Track** button at the bottom of the dialog box. This allows the listing of reports without also displaying the tracks.

23.7.2.1. Trajectory Fates

The possible fates for a particle trajectory are as follows:

- "Escaped" trajectories are those that terminate at a flow boundary for which the "escape" condition is set.
- "Incomplete" trajectories are those that were terminated when the maximum allowed number of time steps — as defined by the **Max. Number of Steps** input in the Discrete Phase Model Dialog Box (p. 2278) (see Numerics of the Discrete Phase Model (p. 1260)) — was exceeded.
- "Incomplete_parallel" may appear as an additional fate for parallel simulations. This means that the number of particle exchanges between partitions has been exceeded. Any remaining particles on the compute nodes are stopped, which is indicated by the number following this fate. Therefore no further source terms from these particles are considered. The number of particle exchanges is limited to avoid very long computational time due to incomplete particles. You can change the default value of 1000 to a value of 20000 with a scheme command. Contact the technical support engineer for this information.
- "Trapped" trajectories are those that terminate at a flow boundary where the "trap" condition has been set.
- "Evaporated" trajectories include those trajectories along which the particles were evaporated within the domain.
- "Aborted" trajectories are those that fail to complete due to roundoff reasons. You may want to retry the calculation with a modified length scale and/or different initial conditions.
- "Shed" trajectories are newly generated particles during the breakup of a larger droplet. They appear only if a breakup model is enabled.
- "Coalesced" trajectories are removed particles which have coalesced after particle-particle collisions. They appear only if the coalescence model is enabled.
- "Splashed" trajectories are particles that are newly generated when a particle touches a wall film. Those trajectories appear only if the wall film model is enabled.

23.7.2.2. Summary Reports

You can request additional detail about the trajectory fates as the particles exit the domain, including the mass flow rates through each boundary zone, mass flow rate of evaporated droplets, and composition of the particles.

1. Follow steps 1 and 2 in Displaying of Trajectories (p. 1330) for displaying trajectories.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select **Summary** as the **Report Type** and click **Display** or **Track**.

> **Important**
>
> For steady-state simulations, DPM summary data is not stored in the `.dat` file, since it is possible to track particles on single or combinations of injections. Transient simulations store this data since it is accumulated over time starting from initialization.

A detailed report similar to the following example will appear in the console window. (You may also choose to write this report to a file by selecting **File** as the **Report to** option, clicking the **Write...** button (which was originally the **Display** button), and specifying a file name for the summary report file in The Select File Dialog Box (p. 19).)

```
  number  tracked  =  10,  escaped  =  3,  aborted  =  0,  trapped  =  5,  evaporated  =  2,

  Fate            Number                   Elapsed  Time  (s)
                                    Min            Max            Avg        Std  Dev
 ----             ------     ----------     ----------     ----------      ----------
Evaporated             2     1.770e-003     1.114e-002     6.456e-003      4.686e-003
Escaped  -  Zone  6     3     6.043e-001     7.037e-001     6.471e-001      4.172e-002
Trapped  -  Zone  7     5     8.486e-003     1.767e-001     5.030e-002      6.421e-002

               (*)-  Mass  Transfer  Summary  -(*)

 Fate                       Mass  Flow  (kg/s)
                          Initial          Final          Change
 ----                  ----------     ----------      ----------
 Evaporated            8.333e-002     0.000e+000     -8.333e-002
 Escaped  -  Zone  6   1.167e-001     5.144e-002     -6.523e-002
 Trapped  -  Zone  7   2.000e-001     2.400e-002     -1.760e-001
 ----                  ----------     ----------      ----------
 Net                   4.000e-001     7.544e-002     -3.246e-001

                (*)-  Energy  Transfer  Summary  -(*)

 Fate                        Heat  Rate  (W)              Change  of  Heat  (W)
                           Initial           Final        Sensible        Latent      React
 ----                   ----------      ----------      ----------     ----------    -------
 Evaporated            -3.180e+004      0.000e+000     -3.382e+002     3.214e+004     1.107
 Escaped  -  Zone  6    5.272e+005      6.519e+005     -3.487e+003     1.282e+005     1.523
 Trapped  -  Zone  7    4.954e+005      6.993e+005     -1.173e+003     2.051e+005     1.737
 ----                   ----------      ----------      ----------     ----------    ------
 Net                    9.908e+005      1.351e+006     -4.998e+003     3.654e+005     4.367

               (*)-  Combusting  Particles  -(*)

 Fate               Volatile  Content  (kg/s)             Char  Content  (kg/s)
                           Initial          Final      %Conv      Initial         Final
 ----                   ----------     ----------    -------   ----------    ----------
 Evaporated             0.000e+000     0.000e+000       0.00   0.000e+000    0.000e+000
 Escaped  -  Zone  6    9.333e-003     9.333e-003       0.00   2.133e-002    2.133e-002
 Trapped  -  Zone  7    9.333e-003     7.485e-010     100.00   2.133e-002    2.133e-002
 ----                   ----------     ----------    -------   ----------    ----------
 Net                    1.867e-002     9.333e-003      50.00   4.267e-002    4.267e-002

               (*)  -  Multicomponent  Droplet  -(*)

 Fate                           Species             Species  Content  (kg/s)
                                  Names          Initial          Final       %Conv
 ----                           -------       ----------     ----------     -------
 Evaporated                 c5h12-droplet<l>      1.667e-002     0.000e+000      100.00
 Evaporated                 c7h16-droplet<l>      3.333e-002     0.000e+000      100.00
 Evaporated                          h2o<l>       0.000e+000     0.000e+000        0.00
 Escaped  -  Zone  6        c5h12-droplet<l>      1.667e-002     2.585e-004       98.45
 Escaped  -  Zone  6        c7h16-droplet<l>      0.000e+000     0.000e+000        0.00
 Escaped  -  Zone  6                 h2o<l>       3.333e-002     1.134e-002       65.99
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Trapped  -  Zone  7     c5h12-droplet<l>      3.333e-002     0.000e+000      100.00
Trapped  -  Zone  7     c7h16-droplet<l>      3.333e-002     0.000e+000      100.00
Trapped  -  Zone  7                h2o<l>      3.333e-002     0.000e+000      100.00
```

The report groups together particles with each possible fate, and reports the number of particles, the time elapsed during trajectories, and the mass and energy transfer This information can be very useful for obtaining information such as where particles are escaping from the domain, where particles are colliding with surfaces, and the extent of heat and mass transfer to/from the particles within the domain. Additional information is reported for combusting particles and multicomponent particles.

23.7.2.2.1. Elapsed Time

The number of particles with each fate is listed under the Number heading. (Particles that escape through different zones or are trapped at different zones are considered to have different fates, and are therefore listed separately.) The minimum, maximum, and average time elapsed during the trajectories of these particles, as well as the standard deviation about the average time, are listed in the Min, Max, Avg, and Std Dev columns. This information indicates how much time the particle(s) spent in the domain before they escaped, aborted, evaporated, or were trapped.

```
Fate          Number                          Elapsed  Time  (s)
                                      Min           Max            Avg       Std  Dev
----               ------      ----------     ----------    ----------      ----------   ---
Incomplete              2       1.485e+01      2.410e+01     1.947e+01       4.623e+00
Escaped  -  Zone  7     8       4.940e+00      2.196e+01     1.226e+01       4.871e+00
```

Also, on the right side of the report are listed the injection name and index of the trajectories with the minimum and maximum elapsed times. (You may need to use the scroll bar to view this information.)

```
  Elapsed  Time  (s)                                  Injection,  Index
Min            Max           Avg      Std  Dev                     Min                 Max
---     ----------    ----------    ----------    --------------------  ------------------
+01      2.410e+01     1.947e+01     4.623e+00        injection-0    1    injection-0    0
+00      2.196e+01     1.226e+01     4.871e+00        injection-0    9    injection-0    2
```

23.7.2.2.2. Mass Transfer Summary

For all droplet or combusting particles with each fate, the total initial and final mass flow rates and the change in mass flow rate are reported in the Initial, Final, and Change columns. With this information, you can determine how much mass was transferred to the continuous phase from the particles.

For unsteady tracking, the report lists the time-integrated mass flow rate of the particle streams that have reached a particular fate at the current flow time. In other words, the report does not include particles that are still being tracked in the domain.

```
       (*)-  Mass  Transfer  Summary  -(*)

Fate                         Mass  Flow  (kg/s)
                       Initial          Final         Change
----                  ----------     ----------     ----------
Incomplete             1.388e-03      1.943e-04      1.194e-03
Escaped  -  Zone  7    1.502e-03      2.481e-04     -1.254e-03
```

23.7.2.2.3. Energy Transfer Summary

This report tells you how much heat was transferred from the particles to the continuous phase. The report is organized in two sections. For steady simulations, there is a Heat Rate and a Change of Heat section. For unsteady particle tracking, there is an Energy and a Change of Energy section. The Heat Rate and Energy sections are the same for all particle types, while the other sections report the change of heat due to the various transfer processes, which differ for each particle type. For steady simulations, the report lists the rate and the change of heat for the particle streams organized according

to the particle stream fates. For unsteady tracking, the report lists the time integrated heat rate and change of the particle streams that have reached a particular fate at the current flow time. Note that the report does not include particles that are still being tracked in the domain.

23.7.2.2.4. Heat Rate and Energy Reporting

For all particles with each fate, the total initial and final heat content are reported in the `Initial` and `Final` columns. The particle heat content H_p is defined as follows:

Inert Particles:

$$H_p = m_p \int_{T_{ref}}^{T_p} C_{p_p} dT \tag{23.14}$$

where:

m_p = mass flow rate of particles (kg/s)

T_p = temperature of particles (K)

C_{p_p} = heat capacity of particles (J/kg/K)

T_{ref} = reference temperature for enthalpy (K)

Droplet Particles:

$$H_p = m_p [f_v \left(-H_{lat_{ref}} + H_{pyrol} \right) + \int_{T_{ref}}^{T_p} C_{p_p} dT] \tag{23.15}$$

where:

H_{pyrol} = heat of pyrolysis (J/kg)

$H_{lat_{ref}}$ = latent heat of evaporation at reference conditions (J/kg)

The latent heat at the reference conditions $H_{lat_{ref}}$ is defined in Equation 16.437 in the Theory Guide.

Combusting Particles:

$$H_p = H_w + H_{dry} \tag{23.16}$$

H_w is the heat content of the evaporating/boiling liquid material if Wet Combustion is selected (otherwise $H_w = 0$).

$$H_w = f_w m_p \left[\left(-H_{lat_{ref}} + H_{pyrol} \right) + \int_{T_{ref}}^{T_p} C_{p_p} dT \right] \tag{23.17}$$

where:

f_w is the mass fraction of the liquid in the combusting particle

$H_{lat_{ref}}$, H_{pyrol}, and C_{p_p} are properties of the evaporating liquid material

H_{dry} is the heat content of the dry combusting particle and is calculated as

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$H_{dry}=\left(1-f_w\right)m_p\left[f_v\left(-H_{lat_{ref}}\right)+f_{comb}H_{comb}+\int_{T_{ref}}^{T_p}C_{p_p}dT\right] \tag{23.18}$$

where:

f_v is the volatile fraction

f_{comb} is the combustible fraction

Important

The Heat Rate section of the report is not provided for the multiple surface reactions model.

Multicomponent Particles:

$$H_p=m_p\sum_i\left(y_iH_{p_i}\right) \tag{23.19}$$

where:

y_i = mass fraction of component i in particle

H_{p_i} = heat content of component i

and

$$H_{p_i}=\left[-H_{lat_{ref_i}}+H_{pyrol_i}+\int_{T_{ref}}^{T_p}C_{p_{p_i}}dT\right] \tag{23.20}$$

where:

H_{pyrol_i} = heat of pyrolysis for component i (J/kg)

$H_{lat_{ref_i}}$ = latent heat of evaporation at reference conditions for component i (J/kg)

$C_{p_{p_i}}$ = specific heat of component i (J/kg/K)

23.7.2.2.4.1. Change of Heat and Change of Energy Reporting

This section reports the total heat transferred from the particle to the continuous phase and is analyzed in components of Sensible heat, Latent heat and heat of Reaction. The Total change reported equals the difference between the Initial and Final states of the particle streams. The sensible heat component is reported for all particle types, the latent heat for the droplet, combusting and multicomponent particle, while the heat of reaction is reported for the combusting particle type only. A positive Change of Heat denotes that heat is expelled from the continuous phase and absorbed by the particle, while a negative Change of Heat denotes heat is released by the particle to the continuous phase.

Steady and Transient Simulations

For steady simulations the report lists the heat rate H_p, while for unsteady tracking the time integrated energy E_p from time 0 to current flow time ts is reported.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$E_p = \int_0^{ts} H_P(t)\,dt \tag{23.21}$$

Below is an example of an Energy Transfer Summary report for evaporating droplets:

```
        (*)-  Energy  Transfer  Summary  -(*)
 Fate                    Heat  Rate  (W)           Change  of  Heat  (W)
                       Initial         Final       Sensible       Latent        Total
 ----                ----------    ----------     ----------   ----------   ----------
 Evaporated         -4.530e+004    0.000e+000    -4.750e+002   4.577e+004   4.530e+004
 Escaped-Zone  6    -1.723e+005   -4.670e+004    -2.552e+003   1.282e+005   1.256e+005
 Trapped-Zone  7    -2.176e+005    0.000e+000    -1.058e+003   2.187e+005   2.176e+005
 ----                ----------    ----------     ----------   ----------   ----------
 Net                -4.353e+005   -4.670e+004    -4.085e+003   3.927e+005   3.886e+005
```

Below is an example of an Energy Transfer Summary report for combusting particles:

```
       (*)-  Energy  Transfer  Summary  -(*)

  Fate                  Heat  Rate  (W)            Change  of  Heat  (W)
                      Initial         Final      Sensible       Latent       Reaction      Tot
  ----                ----------    ----------    ----------    ----------    ----------   ------
 Escaped-Zone  5     1.697e+005    2.555e+004    3.166e+003    1.034e+000    -1.473e+005    -1.44
 Trapped-Zone  6     1.886e+004    1.938e+004    5.731e+002    1.149e-001    -5.370e+001     5.19
  ----                ----------    ----------    ----------    ----------    ----------   ------
  Net                1.886e+005    4.493e+004    3.739e+003    1.149e+000    -1.474e+005    -1.43
```

Important

In a coupled calculation, for all types of steady flows, the Total Net Change of Heat reported in the Energy Transfer Summary should balance with the opposite of the Sum over all fluid cells of the DPM Sensible Enthalpy Source. If this is not the case, this means that the coupled discrete-continuous phase calculation has not converged, and more DPM phase iterations are required. For more information on coupled calculations, see Performing Trajectory Calculations (p. 1325).

```
                              Sum
  DPM  Sensible  Enthalpy  Source                        (w)
 --------------------------------       --------------------
                           fluid-1                -388937.41
```

23.7.2.2.5. Combusting Particles

If combusting particles are present, ANSYS Fluent will include additional reporting on the volatiles and char converted. These reports are intended to help you identify the composition of the combusting particles as they exit the computational domain.

```
                  (*)-  Combusting  Particles  -(*)
 Fate                  Volatile  Content  (kg/s)           Char  Content  (kg/s)
                           Initial          Final     %Conv       Initial        Final     %Conv
 ----                   ----------     ----------   -------    ----------    ---------   -------
 Incomplete              6.247e-04      0.000e+00    100.00     5.691e-04    0.000e+00    100.00
 Escaped  -  Zone  7     6.758e-04      0.000e+00    100.00     6.158e-04    3.782e-05     93.86
```

The total volatile content at the start and end of the trajectory is reported in the Initial and Final columns under Volatile Content. The percentage of volatiles that has been devolatilized is reported in the %Conv column.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The total reactive portion (char) at the start and end of the trajectory is reported in the `Initial` and `Final` columns under `Char Content`. The percentage of char that reacted is reported in the `%Conv` column.

23.7.2.2.6. Combusting Particles with the Multiple Surface Reaction Model

If the multiple surface reaction model is used with combusting particles, ANSYS Fluent will include additional reporting on the mass of the individual solid species that constitute the particle mass.

```
        (*)-  Multiple  Surface  Reactions  -(*)
Fate                  Species       Species  Content  (kg/s)
                       Names       Initial        Final     %Conv
----                 -------    ----------   ----------   -------
Escaped  -  Zone  6     c<s>     6.080e-02    1.487e-06    100.00
Escaped  -  Zone  6     s<s>     3.200e-03    5.077e-06     99.84
Escaped  -  Zone  6      cao     0.000e+00    1.153e-03      0.00
Escaped  -  Zone  6    caso4     0.000e+00    9.266e-04      0.00
Escaped  -  Zone  6    caco3     8.000e-03    5.260e-03     34.25
```

The total mass of each solid species in the particles at the start and end of the trajectory is reported in the `Initial` and `Final` columns, respectively. The percentage of each species that is reacted is reported in the `%Conv` column. Note that for the solid reaction products (for example, if the mass of a solid species has increased in the particle), the conversion is reported to be 0.

23.7.2.2.7. Multicomponent Particles

If your simulation includes multicomponent particles, ANSYS Fluent generates an additional report for the particle components.

```
              (*)  -  Multicomponent  Droplet  -(*)
Fate                              Species            Species  Content  (kg/s)
                                   Names        Initial         Final      %Conv
----                              -------     ----------    ----------    -------
Evaporated                 c5h12-droplet<l>     1.667e-002    0.000e+000     100.00
Evaporated                 c7h16-droplet<l>     3.333e-002    0.000e+000     100.00
Evaporated                          h2o<l>     0.000e+000    0.000e+000       0.00
Escaped  -  Zone  6        c5h12-droplet<l>     1.667e-002    2.585e-004      98.45
Escaped  -  Zone  6        c7h16-droplet<l>     0.000e+000    0.000e+000       0.00
Escaped  -  Zone  6                 h2o<l>     3.333e-002    1.134e-002      65.99
Trapped  -  Zone  7        c5h12-droplet<l>     3.333e-002    0.000e+000     100.00
Trapped  -  Zone  7        c7h16-droplet<l>     3.333e-002    0.000e+000     100.00
Trapped  -  Zone  7                 h2o<l>     3.333e-002    0.000e+000     100.00
```

23.7.3. Step-by-Step Reporting of Trajectories

At times, you may want to obtain a detailed, step-by-step report of the particle trajectory/trajectories. Such reports can be obtained in alphanumeric format. This capability allows you to monitor the particle position, velocity, temperature, or diameter as the trajectory proceeds.

The procedure for generating files containing step-by-step reports is listed below:

1. Follow steps 1 and 2 in Displaying of Trajectories (p. 1330) **Track Single Particle Stream** option.
2. Select **Step by Step** as the **Report Type**.

Important

This option is only available for steady-state cases. For transient cases, see Reporting of Current Positions for Unsteady Tracking (p. 1349).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select **File** as the **Report to** option. (The **Display** button will become the **Write...** button.)

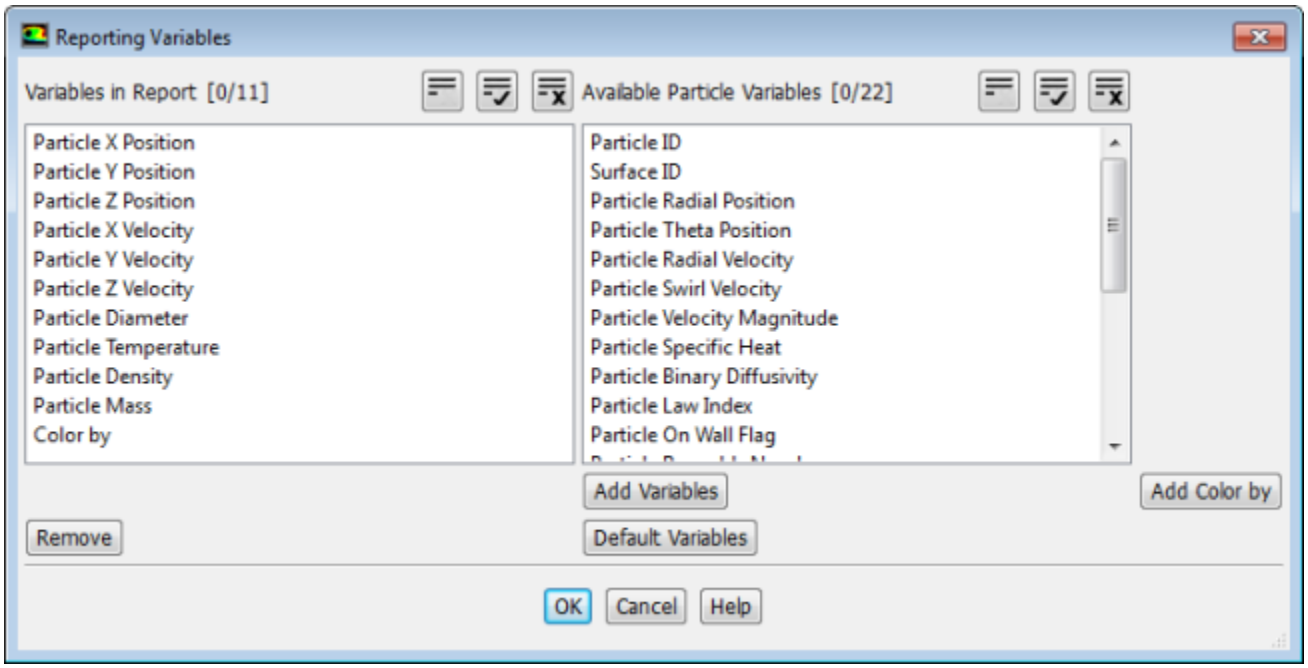
4. In the **Significant Figures** field, enter the number of significant figures to be used in the step-by-step report.

5. Click the **Reporting Variables...** button. The **Reporting Variables** dialog box will appear (Figure 23.46: The Reporting Variables Dialog Box (p. 1348)), where you can change the variables in the report. The list under **Variables in Report** contains all variables currently reported. The **Remove** button removes selected variables from that list. The **Default Variables** button restores the default list. The list under **Particle Variables** contains the particle variables that are available for you to select. You can add selections to the report using the **Add Variable** button. Clicking the **Add Color by** button adds the **Color by** variable to the list of **Variables in Report**, which is the only way to get cell values or customized field functions into the report.

Important

Note that it is possible to select one variable in the **Reporting Variables** dialog box and a variable from the **Color by** drop-down list in the **Particle Tracks** dialog box, however, each variable is reported only once.

Figure 23.46: The Reporting Variables Dialog Box



6. Click the **Write...** button and specify a file name for the step-by-step report file in The Select File Dialog Box (p. 19).

A detailed report similar to the following example will be saved to the specified file before the trajectories are plotted. (You may also choose to print the report in the console by choosing **Console** as the **Report to** option and clicking **Display** or **Track**, but the report is very long that it is unlikely to be of use to you in that form.)

```
FILE  TYPE: 1
COLUMNS:  11
TITLE: TRACK  HISTORY

COLUMN  TYPE  VARIABLE                        (UNITS)
------  ----  --------                        -------
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
      1     2  ParticleResidenceTime            (s)
      2    10  ParticleXPosition                (m)
      3    10  ParticleYPosition                (m)
      4    10  ParticleZPosition                (m)
      5    10  ParticleXVelocity                (m/s)
      6    10  ParticleYVelocity                (m/s)
      7    10  ParticleZVelocity                (m/s)
      8    10  ParticleDiameter                 (m)
      9    10  ParticleTemperature              (K)
     10    10  ParticleDensity                  (kg/m3)
     11    10  ParticleMass                     (kg)
------------------------------------------------

  0.00000e+00   5.00000e-02   5.00000e-02   5.00000e-02   2.00000e+01   0.0000  .  .  .
  1.07087e-07   5.00000e-02   5.00000e-02   5.00000e-02   1.23339e+01   2.6696  .  .  .
  2.51617e-07   5.00000e-02   5.00000e-02   5.00000e-02   1.04417e+01   3.3286  .  .  .
 .              .             .             .             .             .
 .              .             .             .             .             .
 .              .             .             .             .             .
```

The default step-by-step report lists the position, velocity, diameter, temperature, density and mass of the particle at selected time steps along the trajectory. In addition, the variable you have selected in the **Color by** list is also included. If you choose **Console** as the **Report to** option, the variable names are written as the header of each column. (You may need to use the scroll bar to view all variables in this column.)

```
Time       X-Position   Y-Position   Z-Position    X-Velocity   Y-Velocity    Z-Veloc
0.000e+00  1.000e-03    3.120e-02    0.000e+00    1.000e+01     5.000e+00     0.000e
1.672e-05  1.168e-03    3.128e-02    0.000e+00    1.010e+01     4.988e+00     0.000e
3.342e-05  1.337e-03    3.137e-02    0.000e+00    1.019e+01     4.977e+00     0.000e
5.010e-05  1.508e-03    3.145e-02    0.000e+00    1.028e+01     4.965e+00     0.000e
6.675e-05  1.680e-03    3.153e-02    0.000e+00    1.038e+01     4.954e+00     0.000e
8.338e-05  1.854e-03    3.161e-02    0.000e+00    1.047e+01     4.942e+00     0.000e
.          .            .            .            .             .             .
.          .            .            .            .             .             .
.          .            .            .            .             .             .
```

If you change the reporting variables, only those selected will appear in the report. The particle time is always reported in the first column. Note that it is possible to select one variable in the **Reporting Variables** dialog box and a variable from the **Color by** drop-down list in the **Particle Tracks** dialog box, however, each variable is reported only once.

When the report is written to a file, a table at the beginning of the file lists all variables selected with the corresponding unit. Thus you can display or export any variable along a particle trajectory to the console or to a file.

Note that the **Coarsen** option affects the step-by-step report.

23.7.4. Reporting of Current Positions for Unsteady Tracking

In transient cases, when using unsteady tracking, you may want to obtain a report of the particle trajectory/trajectories showing the current positions of the particles. Selecting **Current Positions** under **Report Type** in the Particle Tracks Dialog Box (p. 2761) enables the display of the current positions of the particles.

The procedure for generating files containing current position reports is listed below:

1. Follow steps 1 and 2 in Displaying of Trajectories (p. 1330) for displaying trajectories. You may want to track only one particle stream at a time, using the **Track Single Particle Stream** option.

2. Select **Current Position** as the **Report Type**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select **File** as the **Report to** option. (The **Display** button will become the **Write...** button.)

4. In the **Significant Figures** field, enter the number of significant figures to be used in the report.

5. Click the **Reporting Variables...** button. The **Reporting Variables** dialog box will appear (Figure 23.46: The Reporting Variables Dialog Box (p. 1348)), where you can change the variables in the report. The list under **Variables in Report** contains all variables currently reported. The **Remove** button removes selected variables from that list. The **Default Variables** button restores the default list. The list under **Particle Variables** contains the particle variables that are available for you to select. You can add selections to the report using the **Add Variable** button. Clicking the **Add Color by** button adds the **Color by** variable to the list of **Variables in Report**, which is the only way to get cell values or customized field functions into the report.

   Important

   Note that it is possible to select one variable in the **Reporting Variables** dialog box and a variable from the **Color by** drop-down list in the **Particle Tracks** dialog box, however, each variable is reported only once.

6. Click the **Write...** button and specify a file name for the current position report file in The Select File Dialog Box (p. 19).

The default current position report lists the position, velocity, diameter, temperature, density, mass and number in parcel of the particle at selected time steps along the trajectory. In addition, the variable you have selected in the **Color by** list is also included. If you change the reporting variables, only those selected will appear in the report. The particle time is always reported in the first column. It is possible to select one variable in the **Reporting Variables** dialog box and a variable from the **Color by** drop-down list in the **Particle Tracks** dialog box, however, each variable is reported only once.

The output to a file or to the console has the same format as the step-by-step report for steady-state cases.

```
   Time         X-Position    Y-Position    Z-Position   X-Velocity    Y-Velocity    Z-Velocity
0.000e+00     1.000e-03     3.120e-02     0.000e+00     1.000e+01     5.000e+00     0.000e+00
1.672e-05     1.168e-03     3.128e-02     0.000e+00     1.010e+01     4.988e+00     0.000e+00
3.342e-05     1.337e-03     3.137e-02     0.000e+00     1.019e+01     4.977e+00     0.000e+00
5.010e-05     1.508e-03     3.145e-02     0.000e+00     1.028e+01     4.965e+00     0.000e+00
6.675e-05     1.680e-03     3.153e-02     0.000e+00     1.038e+01     4.954e+00     0.000e+00
8.338e-05     1.854e-03     3.161e-02     0.000e+00     1.047e+01     4.942e+00     0.000e+00
 .             .             .             .             .             .             .
 .             .             .             .             .             .             .
 .             .             .             .             .             .             .
```

Also listed are the diameter, temperature, density, mass of the particles, number in parcel and the variable selected from the **Color by** list. (You may need to use the scroll bar to view this information.)

```
   Time         Diameter    Temperature    Density         Mass          Number      ColorBy
 9.999e-04     9.352e-05    3.710e+02    6.840e+02     2.929e-10     2.792e+02    4.783e-02
 1.999e-03     7.952e-05    3.710e+02    6.840e+02     1.801e-10     2.792e+02    3.834e-02
 3.000e-03     6.660e-05    3.710e+02    6.840e+02     1.058e-10     2.792e+02    2.989e-02
 4.001e-03     5.425e-05    3.710e+02    6.840e+02     5.719e-11     2.792e+02    3.719e-02
 5.001e-03     4.184e-05    3.710e+02    6.840e+02     2.624e-11     2.792e+02    2.978e-02
  .             .            .            .             .             .            .
  .             .            .            .             .             .            .
  .             .            .            .             .             .            .
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23.7.5. Reporting of Interphase Exchange Terms (Discrete Phase Sources)

ANSYS Fluent reports the magnitudes of the interphase exchange of momentum, heat, and mass in each control volume in your ANSYS Fluent model. You can display these variables graphically, by drawing contours, profiles, and so on. They are all contained in the **Discrete Phase Sources...** category of the variable selection drop-down list that appears in postprocessing dialog boxes:

- **DPM Mass Source**
- **DPM X,Y,Z Momentum Source**
- **DPM Swirl Momentum Source**
- **DPM Turbulent Kinetic Energy Source**
- **DPM Turbulent Dissipation Source**
- **DPM Sensible Enthalpy Source**
- **DPM Enthalpy Source**
- **DPM Burnout**
- **DPM Evaporation/Devolatilization**
- **DPM (species) Source**

See Field Function Definitions (p. 1931) for definitions of these variables.

Note that these exchange terms are updated and displayed only when coupled calculations are performed. Displaying and reporting particle trajectories (as described in Displaying of Trajectories (p. 1330) and Reporting of Trajectory Fates (p. 1340)) will not affect the values of these exchange terms.

The exchange terms are reported as the rate occurring in each cell. A unit cell depth is used for 2D cases, and a reference cell depth of 1 radian is used for 2D axisymmetric cases.

23.7.6. Reporting of Discrete Phase Variables

ANSYS Fluent reports various discrete phase particle/parcel quantities including erosion/accretion rates, radiation quantities, and (optionally) cell-averaged particle size, velocity, temperature, and so on. You can display these variables graphically, by drawing contours, profiles, and so on. They are accessed in the **Discrete Phase Variables...** category of the variable selection drop-down list that appears in postprocessing dialog boxes.

Several quantities are automatically available depending on the models being used in the simulation. For the cell-averaged quantities to be available, you must first enable **Mean Values** under **Contour Plots for DPM Variables** in the **Discrete Phase Model** dialog box, Discrete Phase Model Dialog Box (p. 2278). RMS quantities are also available for particle velocity and temperature by enabling **RMS Values**.

Note

Enabling Mean Values to track the cell-averaged variables will increase the memory requirements of your simulation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following lists specify those variables that are available automatically and those that require you to enable them

Variables available automatically

- **DPM Erosion Rate (Generic)**
- **DPM Erosion Rate (Finnie)**
- **DPM Erosion Rate (McLaury)**
- **DPM Erosion Rate (Oka)**
- **DPM Accretion Rate**
- **DPM Absorption Coefficient**
- **DPM Emission**
- **DPM Scattering**
- **DPM Concentration**
- **DPM (species) Concentration**
- **DPM Collision Rate**

Particle rotation variables (Particle rotation simulations only)

- **Angular X Velocity**
- **Angular Y Velocity**
- **Angular Z Velocity**
- **Angular Velocity Magnitude**
- **Angular Velocity** (in the **Particle Vector Style** dialog box)

Cell-averaged variables available when Mean Values are enabled

- **DPM Volume Fraction**
- **DPM Particles in Cell**
- **DPM Parcels in Cell**
- **DPM Number Density**
- **DPM X, Y, Z Velocity**
- **DPM Diameter**
- **DPM Density**
- **DPM Temperature**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **DPM Specific Heat**
- **DPM D20**
- **DPM D30**
- **DPM D32**
- **DPM Granular Temperature**
- **DPM Conc. of (component)**

RMS variables available when Mean Values and RMS Values are enabled

- **DPM RMS X, Y, Z Velocity**
- **DPM RMS Temperature**
- **DPM RMS Diameter**

See Field Function Definitions (p. 1931) for definitions of these variables.

Note that these variables are updated and displayed only when coupled calculations are performed. Displaying and reporting particle trajectories (as described in Displaying of Trajectories (p. 1330) and Reporting of Trajectory Fates (p. 1340)) will not affect the values of these variables.

23.7.7. Reporting of Unsteady DPM Statistics

If you are performing a transient simulation, you can include the computation of unsteady time statistics (mean and RMS) for the discrete phase(s) of the transient flow. These are computed on a per-phase basis by event-based time averaging over the discrete phase parcels in the domain. This means that the sampling of the discrete phase particle quantity in a given cell occurs when (and only when) a parcel passes through the cell.

To calculate the unsteady DPM time statistics, enable **Data Sampling for Time Statistics** in the **Run Calculation** task page and enable **DPM Variables** in the **Sampling Options** dialog box when preparing to run your simulation (see User Inputs for Time-Dependent Problems (p. 1609)).

For a discrete phase variable, ϕ, the event-based time average in a cell is calculated from Equation 23.22 (p. 1353).

$$\langle \overline{\phi} \rangle = \frac{\sum_{i=1}^{N} \phi_i w_i}{\sum_{i=1}^{N} w_i} \tag{23.22}$$

where N is the number of parcels in the domain.

$$w_i = \frac{n_{p,i} t_i}{T_j} \tag{23.23}$$

Here $n_{p,i}$ is the number of particles in the i^{th} parcel and t_i is the residence time of the i^{th} parcel in the cell. T_j is the computational time step. The weighting by residence time is necessary because a parcel

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

may pass through more than one cell within a computational time step and therefore its contribution to the averaged cell quantities depends on the fraction of the time step spent in each cell.

The RMS value is also available and is computed from Equation 23.24 (p. 1354).

$$\phi_{rms}=\sqrt{\frac{\sum_{i=1}^{N}\left(\phi_i-\langle\overline{\phi}\rangle\right)^2 w_i}{\sum_{i=1}^{N} w_i}} \tag{23.24}$$

Note that these quantities are averaged over all parcels that have passed through the cell and are therefore different from the instantaneous cell-averaged values described in Reporting of Discrete Phase Variables (p. 1351).

Below is a list of the available unsteady statistics when **DPM Variables** is enabled in the **Sampling Options** dialog box. These are accessible by selecting the **Unsteady DPM Statistics...** category in postprocessing or reporting dialog boxes. Note that the availability of some quantities depends on the physics models being used. For definitions of these quantities, refer to the definitions of the instantaneous quantities from which they are derived (see Alphabetical Listing of Field Variables and Their Definitions (p. 1957)).

Unsteady DPM Statistics

- **Mean DPM Volume Fraction**
- **Accum DPM Particles in Cell**
- **Mean DPM X, Y, Z Velocity**
- **Mean DPM Diameter**
- **Mean DPM Density**
- **Mean DPM Temperature**
- **Mean DPM Granular Temperature**
- **Mean DPM Number Density**
- **Accum DPM Parcels in Cell**
- **Mean DPM D20**
- **Mean DPM D30**
- **Mean DPM D32**
- **Mean DPM Sauter Diameter**
- **RMS DPM Volume Fraction**
- **RMS DPM X, Y, Z Velocity**
- **RMS DPM Diameter**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

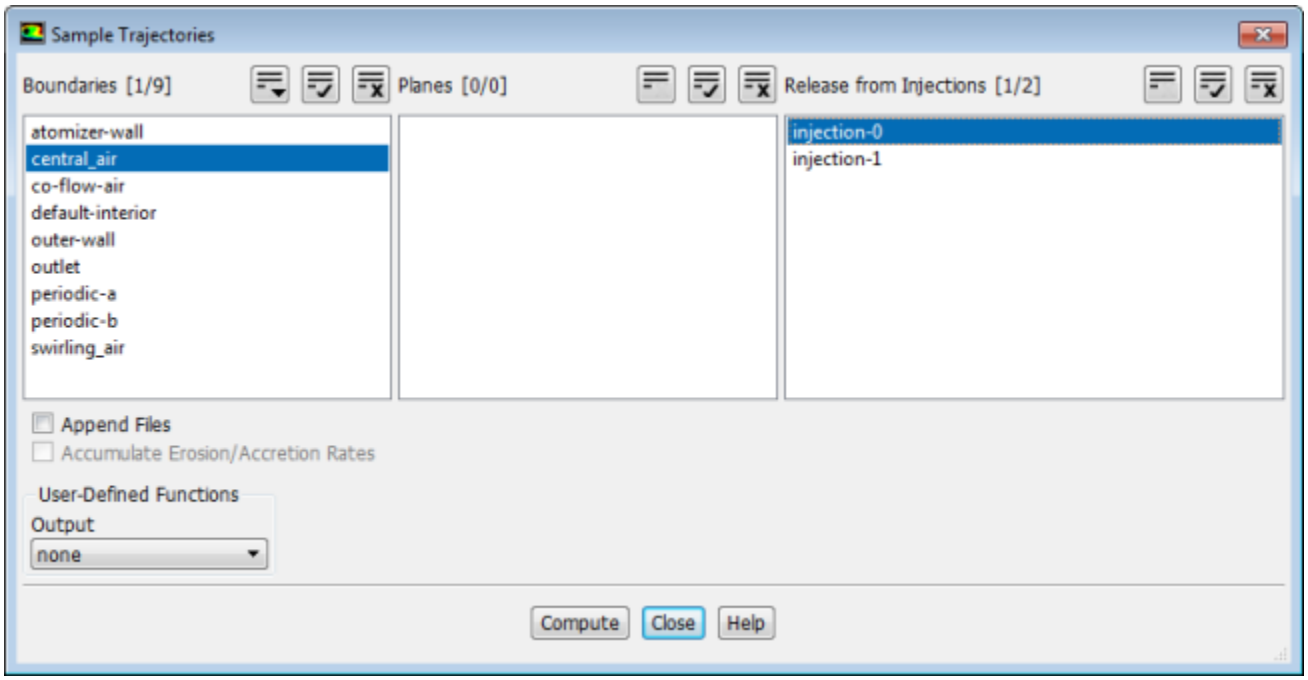
- **RMS DPM Density**
- **RMS DPM Temperature**
- **RMS DPM Granular Temperature**
- **RMS DPM Number Density**

23.7.8. Sampling of Trajectories

Particle states (position, velocity, diameter, temperature, and mass flow rate) can be written to files at various boundaries and planes (lines in 2D) using the Sample Trajectories Dialog Box (p. 2622) (Figure 23.47: The Sample Trajectories Dialog Box (p. 1355)).

Results → Reports → Discrete Phase → Sample → Edit...

Figure 23.47: The Sample Trajectories Dialog Box



The procedure for generating files containing the particle samples is listed below:

1. Select the injections to be tracked in the **Release From Injections** list.
2. Select the surfaces at which samples will be written These can be boundaries from the **Boundaries** list or planes from the **Planes** list (in 3D) or lines from the **Lines** list (in 2D).
3. Click the **Compute** button. Note that for unsteady particle tracking, the **Compute** button will become the **Start** button (to initiate sampling) or a **Stop** button (to stop sampling).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clicking the **Compute** button will cause the particles to be tracked and their status to be written to files when they encounter selected surfaces. The file names will be formed by appending `.dpm` to the surface name.

Note

If you select a face zone in a mesh interface for sampling, Fluent will sample the particles on both of the interface zones and will write out a total of three files. Two of these files correspond to the particles that encounter each respective interface zone from upstream. The third file contains the complete set of particles that have crossed the mesh interface pair (via either interface zone).

For unsteady particle tracking, clicking the **Start** button will open the files and write the file header sections. If the solution is advanced in time by computing some time steps, the particle trajectories will be updated and the particle states will be written to the files as they cross the selected planes or boundaries. Clicking the **Stop** button will close the files and end the sampling.

For stochastic tracking, it may be useful to repeat this process multiple times and append the results to the same file, while monitoring the sample statistics at each update. To do this, enable the **Append Files** option before repeating the calculation (clicking **Compute**). Similarly, you can cause erosion and accretion rates to be accumulated for repeated trajectory calculations by turning on the **Accumulate Erosion/Accretion Rates** option. (See also Postprocessing of Erosion/Accretion Rates (p. 1360).) The format and the information written for the sample output can also be controlled through a user-defined function, which can be selected in the **Output** drop-down list. More information about user-defined functions can be found in the Fluent Customization Manual.

When sampling steady particle tracks the generated sample files can be used as injection file in a file injection. Both files use a similar file format.

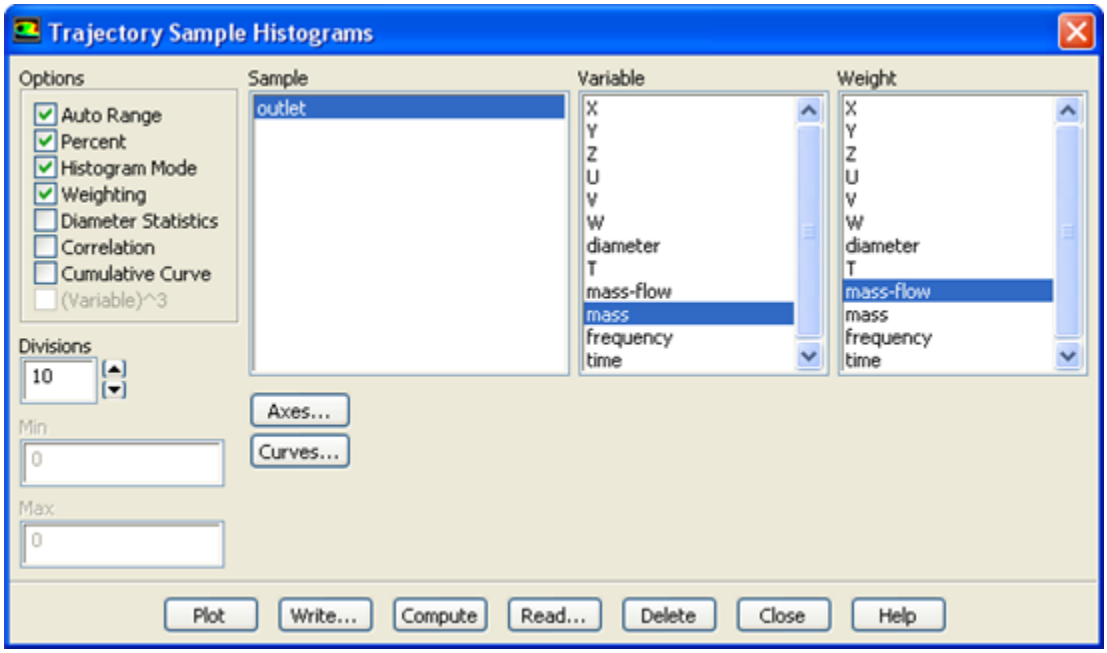
23.7.9. Histogram Reporting of Samples

Histograms can be plotted from sample files created in the Sample Trajectories Dialog Box (p. 2622) (as described in Sampling of Trajectories (p. 1355)) using the Trajectory Sample Histograms Dialog Box (p. 2623) (Figure 23.48: The Trajectory Sample Histograms Dialog Box (p. 1357)).

Results → Reports → Discrete Phase → Histogram Edit...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.48: The Trajectory Sample Histograms Dialog Box



The procedure for plotting histograms from data in a sample file is listed below:

1. Select a file to be read by clicking the **Read...** button. After you read in the sample file, the boundary name will appear in the **Sample** list.

2. Select the data sample in the **Sample** list, and then select the data to be plotted from the **Variable** list.

3. Select the data to weight the variable from the **Weight** list.

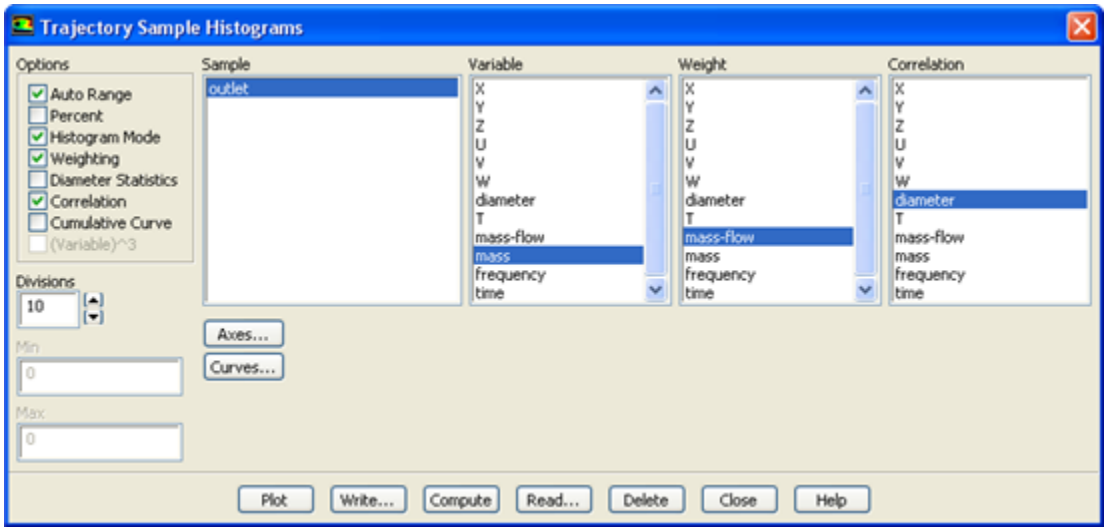
4. Click the **Plot** button at the bottom of the dialog box to display the histogram.

By default, the percent of particles will be plotted on the y axis. You can plot the actual number of particles by deselecting **Percent** under **Options**. The number of "bins" or intervals in the plot can be set in the **Divisions** field. You can delete samples from the list with the **Delete** button and update the **Min/Max** values with the **Compute** button. To display the histogram without the bars, deselect **Histogram Mode** under **Options**. A summary similar to that in Summary Reporting of Current Particles (p. 1358) will be displayed in the console for the selected variables when **Diameter Statistics** is enabled. Although these statistics are computed for the selected variable in the **Variable** list, it is applicable only to the diameter information. When the sampled particle streams all have the same flow rate, you can disable the **Weighting**. To postprocess the histograms with other tools, you can store them in an XY-plot file format using the **Write...** button.

When investigating the behavior of particles, it is sometimes desirable to know how one type of particle variable depends on another particle variable. To facilitate this, the **Correlation** option exists. When you enable this option, an additional column of sampled variables appears, allowing you to choose the correlation variable (see Figure 23.49: The Trajectory Sample Histograms Dialog Box (p. 1358)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.49: The Trajectory Sample Histograms Dialog Box



If you want to know the continuous cumulative distributions, enable the **Cumulative Curve** option. A cumulative distribution curve is computed of the variable that is selected in the **Variable** list. However, if the **Correlation** option is enabled along with the **Cumulative Curve**, then the cumulative curve of the variable selected in the **Correlation** list is plotted. For a constant particle density, you can plot the cumulative mass distribution by selecting the diameter and enabling `(Variable)^3`.

23.7.10. Summary Reporting of Current Particles

For many mass-transfer and flow processes, it is desirable to know the mean diameter of the particles. A mean diameter, D_{jk}, is calculated from the particle size distribution using the following general expression [55] (p. 2890):

$$\left(D_{jk}\right)^{j-k} \equiv \frac{\int_0^\infty D^j f(D)\,dD}{\int_0^\infty D^k f(D)\,dD} \tag{23.25}$$

where j and k are integers and $f(D)$ is the distribution function (for example, Rosin-Rammler). D_{10}, for example, is the average (arithmetic) particle diameter. The Sauter mean diameter (SMD), D_{32}, is the diameter of a particle whose ratio of volume to surface area is equal to that of all particles in the computation. A summary of common mean diameters is given in Table 23.6: Common Mean Diameters and Their Fields of Application (p. 1358).

Table 23.6: Common Mean Diameters and Their Fields of Application

j	k	**Order** $j+k$	**Name**	**Field of Application**
1	0	1	Mean diameter, D_{10}	Comparisons, evaporation
2	0	2	Mean surface diameter, D_{20}	Absorption
3	0	3	Mean volume diameter, D_{30}	Hydrology
2	1	3	Overall surface diameter, D_{21}	Adsorption

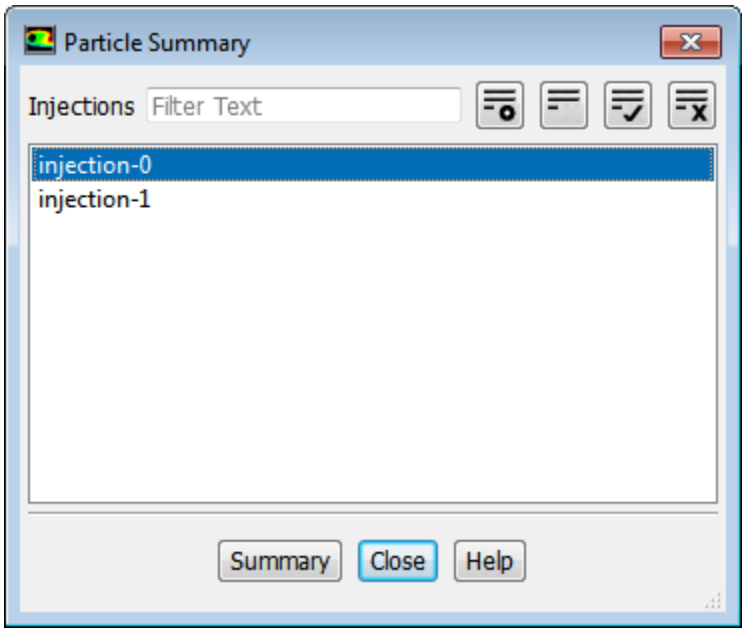
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

j	k	**Order** $j+k$	**Name**	**Field of Application**
3	1	4	Overall volume diameter, D_{31}	Evaporation, molecular diffusion
3	2	5	Sauter mean diameter, D_{32}	Combustion, mass transfer, and efficiency studies
4	3	7	De Brouckere diameter, D_{43}	Combustion equilibrium

Summary information (number, mass, average diameter) for particles currently in the computational domain can be reported using the Particle Summary Dialog Box (p. 2625) (Figure 23.50: The Particle Summary Dialog Box (p. 1359))

Results → Reports → Discrete Phase → Summary Edit...

Figure 23.50: The Particle Summary Dialog Box



The procedure for reporting a summary for particle injections is as follows:

1. Select the particle injection(s) you want summarized in the **Injections** list.

   Fluent provides a shortcut for selecting injections with names that match a specified pattern. To use this shortcut, enter the pattern in the **Filter Text** box. For example, if you specify `drop*`, all injections that have names beginning with **drop** (for example, **drop-1**, **droplet**) will be listed.

2. Click **Summary** to display the injection summary in the console window.

```
 (*)- Summary for Injection: injection-0 -(*)

Total number of parcels             :  1862
Total number of particles           :  1.196710e+05
Total mass                          :  1.128303e-05 (kg)
Maximum RMS distance from injector  :  7.372527e-01 (m)
Maximum particle diameter           :  3.072739e-04 (m)
Minimum particle diameter           :  1.756993e-06 (m)
Overall RR Spread Parameter         :  1.446806e+00
Maximum Error in RR fit             :  1.071220e-01
```

```
Overall  RR  diameter            (D_RR):  9.051303e-05 (m)
Overall  mean  diameter          (D_10):  4.663269e-05 (m)
Overall  mean  surface  area     (D_20):  5.344694e-05 (m)
Overall  mean  volume            (D_30):  6.121478e-05 (m)
Overall  surface  diameter       (D_21):  6.125692e-05 (m)
Overall  volume  diameter        (D_31):  7.013570e-05 (m)
Overall  Sauter  diameter        (D_32):  8.030141e-05 (m)
Overall  De  Brouckere  diameter  (D_43):  1.082971e-04 (m)
```

23.7.11. Postprocessing of Erosion/Accretion Rates

You can calculate the erosion and accretion rates in a cumulative manner (over a series of injections) by using the **Sample Trajectories** dialog box. First select an injection in the **Release From Injections** list and compute its trajectory. Then enable the **Accumulate Erosion/Accretion Rates** option, select the next injection (after deselecting the first one), and click **Compute** again. The rates will accumulate at the surfaces each time you click **Compute**.

Important

Both the erosion rate and the accretion rate are defined at wall face surfaces only, so they cannot be displayed at node values.

23.8. Parallel Processing for the Discrete Phase Model

ANSYS Fluent offers three modes of parallel processing for the discrete phase model:

- **Shared Memory**

  The **Shared Memory** method is suitable for computations where the machine running the ANSYS Fluent host process is an adequately large, shared-memory, multiprocessor machine.

- **Message Passing**

  The **Message Passing** option is suitable for generic distributed memory cluster computing.

- **Hybrid** (default)

  The **Hybrid** option is suitable for modern multicore memory cluster computing.

These options are found under the **Parallel** tab, in the **Discrete Phase Model** dialog box.

Important

When tracking particles in parallel, the DPM model cannot be used with any of the multiphase flow models (VOF, mixture, or Eulerian) if the **Shared Memory** option is enabled. (Note that using the **Message Passing** or **Hybrid** option, when running in parallel, enables the compatibility of all multiphase flow models with the DPM model.)

If you want to use **file** injections or to write a DPM summary report directly to a file, your system must meet the following requirements when either the **Message Passing** or **Hybrid** parallel DPM tracking option is selected:

- The machine (for example, cluster node) running the compute-node zero process and the machine running the host process are identical or have the same operating system type (Windows or Linux)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Both machines can access the same file system through the same path specification

If these conditions are not met, you can still write a DPM summary report to a file by starting transcript, reporting the particle tracking summary to the Fluent console, and then stopping transcript. The summary report will be saved in the transcript file.

As a workaround for the DPM summary report, you can start transcript, report the particle tracking summary to the Fluent console, and stop transcript. The summary report will be saved in the transcript file

The **Shared Memory** option (Figure 23.51: The Shared Memory Option with Workpile Algorithm Enabled (p. 1362)) is implemented using POSIX Threads (pthreads) based on a shared-memory model. Once the **Shared Memory** option is enabled, you can then select along with it the **Workpile Algorithm** and specify the **Number of Threads**. By default, the **Number of Threads** is equal to the number of compute nodes specified for the parallel computation. You can modify this value based on the computational requirements of the particle calculations. If, for example, the particle calculations require more computation than the flow calculation, you can increase the **Number of Threads** (up to the number of available processors) to improve performance. When using the **Shared Memory** option, the particle calculations are entirely managed by the ANSYS Fluent host process. You must make sure that the machine executing the host process has enough memory to accommodate the entire mesh.

Important

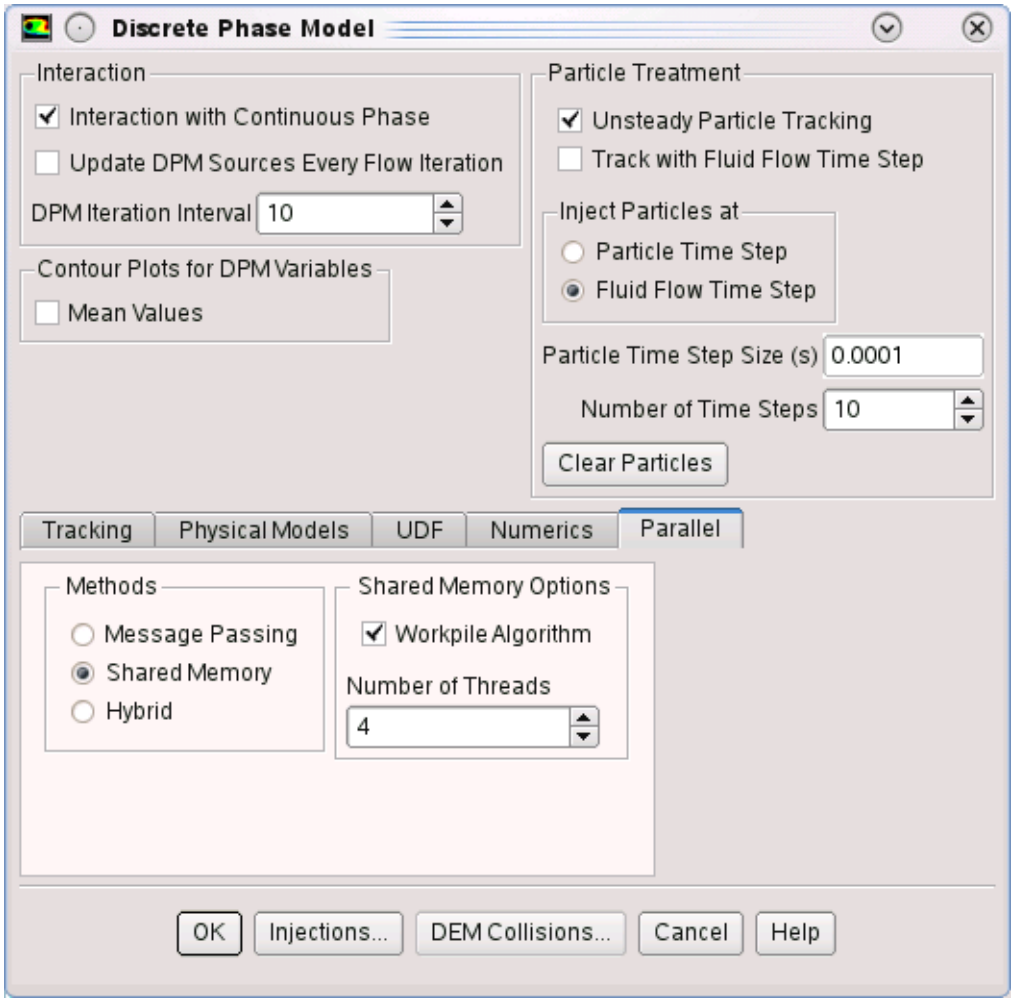
Note that the **Shared Memory** option on Windows based architectures only provides serial tracking on the host, since the **Workpile Algorithm** is not available due to lack of POSIX Threads on these platforms.

Important

Note that the **Workpile Algorithm** option is not available with the wall film boundary condition. It will be disabled automatically when choosing to simulate a wall film on a wall.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.51: The Shared Memory Option with Workpile Algorithm Enabled



The **Message Passing** option enables cluster computing and also works on shared-memory machines. With this option enabled, the compute node processes perform the particle work on their local partitions. Particle migration to other compute nodes is implemented using message passing primitives. There are no special requirements for the host machine. Note that this model is not available if the **Cloud Model** option is turned on under the **Turbulent Dispersion** tab of the **Set Injection Properties** dialog box. When running ANSYS Fluent in parallel, by default, pathline displays are computed in serial on the host node. Pathline displays may be computed in parallel on distributed memory systems if the **Message Passing** parallel option is selected in the **Discrete Phase Model** dialog box.

The **Hybrid** option combines **Message Passing** and **OpenMP** for a dynamic load balancing without migration of cells, enables multicore cluster computing, and also works on shared-memory machines. With this option enabled, the compute node processes perform the particle calculations on their local partitions. **OpenMP** threads will be spawned, and the number of threads in each ANSYS Fluent node process is based on the evaluation of the particle load on the current machine. The maximum number of threads on each machine can be controlled using the **Thread Control** dialog box (see Controlling the Threads (p. 2058) for details). The default value is the number of ANSYS Fluent node processes on each machine. Particle migration to other compute nodes is implemented using message passing primitives. There are no special requirements for the host machine. Note that this model is not available if the **Cloud Model** option is turned on under the **Turbulent Dispersion** tab of the **Set Injection Properties** dialog box. When running ANSYS Fluent in parallel, pathline displays are computed by default in serial on the host node. Pathline displays may also be computed in parallel on distributed memory

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

systems if the **Message Passing** option is selected from the **Methods** list. For a list of limitations that exist with Hybrid parallel method, see Limitations on Using the Hybrid Parallel Method (p. 1239).

When using the Hybrid method, you may optionally enable **Use DPM Domain** under **Hybrid Options**. This option can provide substantially improved load balancing, and thus scalability, at the expense of additional memory overhead. When **Use DPM Domain** is enabled, particle tracking is performed on a separate domain from the root computational domain. Flow and DPM variables are copied between the domains as part of the solution process. By using a separate domain, and thus partitioning strategy, for particle tracking the continuous and discrete phase loads are balanced independently. This allows the load to be shared more equally among the machines regardless of how particles are distributed throughout the computational domain. This is especially beneficial for simulations with non-uniform particle distributions over the computational domain. For limitations associated with the **Use DPM Domain** option, see Limitations on Using the Hybrid Parallel Method (p. 1239). To enable dynamic load balancing with this option, use the following text command:

`parallel` → `partition` → `set` → `dpm-load-balancing`

When this load balancing is enabled, you will be prompted to define a load imbalance threshold (that is, a percentage above which load balancing will occur), as well as the interval of DPM iterations at which you would like load balancing attempted. It is recommended that you retain the default values, as these are appropriate for a wide range of cases.

You may seamlessly switch among the **Shared Memory** option, the **Message Passing** option, and the **Hybrid** option at any time during the ANSYS Fluent session.

In addition to performing general parallel processing of the Discrete Phase Model, you have the option of implementing DPM-specific user-defined functions in parallel ANSYS Fluent. For more information about the parallelization of DPM UDFs, see Parallelization of Discrete Phase Model (DPM) UDFs in the Fluent Customization Manual.

When using the **Message Passing** or the **Hybrid** option you can make use of ANSYS Fluent's automated load balancing capability by giving an appropriate weight to the particle steps in each cell. In this case the number of particle steps in each partition is considered in the load balancing procedure. Further details can be found in Using the Partitioning and Load Balancing Dialog Box (p. 2033).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 24: Modeling Multiphase Flows

This chapter discusses the general multiphase models that are available in ANSYS Fluent. For information about the various theories behind the general multiphase models in ANSYS Fluent, see Multiphase Flows in the Theory Guide. Information about using the general multiphase models in ANSYS Fluent is presented in the following sections:

24.1. Introduction
24.2. Steps for Using a Multiphase Model
24.3. Setting Up the VOF Model
24.4. Setting Up the Mixture Model
24.5. Setting Up the Eulerian Model
24.6. Setting Up the Wet Steam Model
24.7. Solution Strategies for Multiphase Modeling
24.8. Multiphase Case Check
24.9. Postprocessing for Multiphase Modeling

24.1. Introduction

The first step in solving any multiphase problem is to determine which of the regimes described in Multiphase Flow Regimes in the Theory Guide best represents your flow. Model Comparisons in the Theory Guide provides some broad guidelines for determining appropriate models for each regime, and Detailed Guidelines provides details about how to determine the degree of interphase coupling for flows involving bubbles, droplets, or particles, and the appropriate model for different amounts of coupling.

The following sections will guide you through the setup, solution, and postprocessing of multiphase flow models.

24.2. Steps for Using a Multiphase Model

The procedure for setting up and solving a general multiphase problem is outlined below, and described in detail in the subsections that follow. Remember that only the steps that are pertinent to general multiphase calculations are shown here. For information about inputs related to other models that you are using in conjunction with the multiphase model, see the appropriate sections for those models.

See also Additional Guidelines for Eulerian Multiphase Simulations (p. 1455) for guidelines on simplifying Eulerian multiphase simulations.

Important

Double Precision is recommended for all multiphase cases.

1. Enable the multiphase model you want to use (VOF, mixture, or Eulerian) and specify the number of phases. For the VOF and Eulerian models, specify the volume fraction scheme as well.

   Setup → **Models** → **Multiphase** → **Edit...**

See Enabling the Multiphase Model (p. 1367) and Choosing Volume Fraction Formulation (p. 1371) for details.

2. Copy the material representing each phase from the materials database.

   Setup → Materials

   If the material you want to use is not in the database, create a new material. See Using the **Create/Edit Materials** Dialog Box (p. 461) for details about copying from the database and creating new materials. See Modeling Compressible Flows (p. 1375) for additional information about specifying material properties for a compressible phase. It is possible to turn off reactions in some materials by selecting **none** in the **Reactions** drop-down list under **Properties** in the **Create/Edit Materials** dialog box.

   Important

   If your model includes a particulate (granular) phase, you will need to create a new material for it in the *fluid* materials category (*not* the solid materials category).

3. Define the phases, and specify any interaction between them (for example, surface tension if you are using the VOF model, slip velocity functions if you are using the mixture model, or drag functions if you are using the Eulerian model).

   Setup → Models → Multiphase → Phases → Edit...

   See Defining the Phases (p. 1377) – Defining the Phases for the Eulerian Model (p. 1456) for details.

4. (Eulerian model only) If the flow is turbulent, define the multiphase turbulence model.

   Setup → Models → Viscous → Edit...

   See Modeling Turbulence (p. 1474) for details.

5. If body forces are present, enable gravity and specify the gravitational acceleration.

   Setup → Cell Zone Conditions → Operating Conditions...

   See Including Body Forces (p. 1377) for details.

6. Specify the boundary conditions, including the secondary-phase volume fractions at flow boundaries and (if you are modeling wall adhesion in a VOF simulation) the contact angles at walls.

   Setup → Boundary Conditions

   See Defining Multiphase Cell Zone and Boundary Conditions (p. 1391) for details.

7. Set any model-specific solution parameters.

   Solution → Methods

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution → **Controls**

See Setting Time-Dependent Parameters for the Explicit Volume Fraction Formulation (p. 1445) and Solution Strategies for Multiphase Modeling (p. 1504) for details.

8. Initialize the solution and set the initial volume fractions for the secondary phases.

   Solution → **Initialization** **Patch...**

   See Setting Initial Volume Fractions (p. 1407) for details.

9. Calculate a solution and examine the results. Postprocessing and reporting of results are available for each phase that is selected.

   See Solution Strategies for Multiphase Modeling (p. 1504) and Postprocessing for Multiphase Modeling (p. 1520) for details.

Note

For information on multiphase model compatibility with other Fluent models, see Appendix A: ANSYS Fluent Model Compatibility (p. 2847).

This section provides instructions and guidelines for using the VOF, mixture, and Eulerian multiphase models.

Information is presented in the following subsections:

24.2.1. Enabling the Multiphase Model
24.2.2. Choosing Volume Fraction Formulation
24.2.3. Solving a Homogeneous Multiphase Flow
24.2.4. The Boussinesq Approximation in Multiphase Flow
24.2.5. Modeling Compressible Flows
24.2.6. Defining the Phases
24.2.7. Including Body Forces
24.2.8. Modeling Multiphase Species Transport
24.2.9. Specifying Heterogeneous Reactions
24.2.10. Including Mass Transfer Effects
24.2.11. Defining Multiphase Cell Zone and Boundary Conditions
24.2.12. Setting Initial Volume Fractions

24.2.1. Enabling the Multiphase Model

To enable the VOF, mixture, or Eulerian multiphase model, select **Volume of Fluid**, **Mixture**, or **Eulerian** under **Model** in the Multiphase Model Dialog Box (p. 2171).

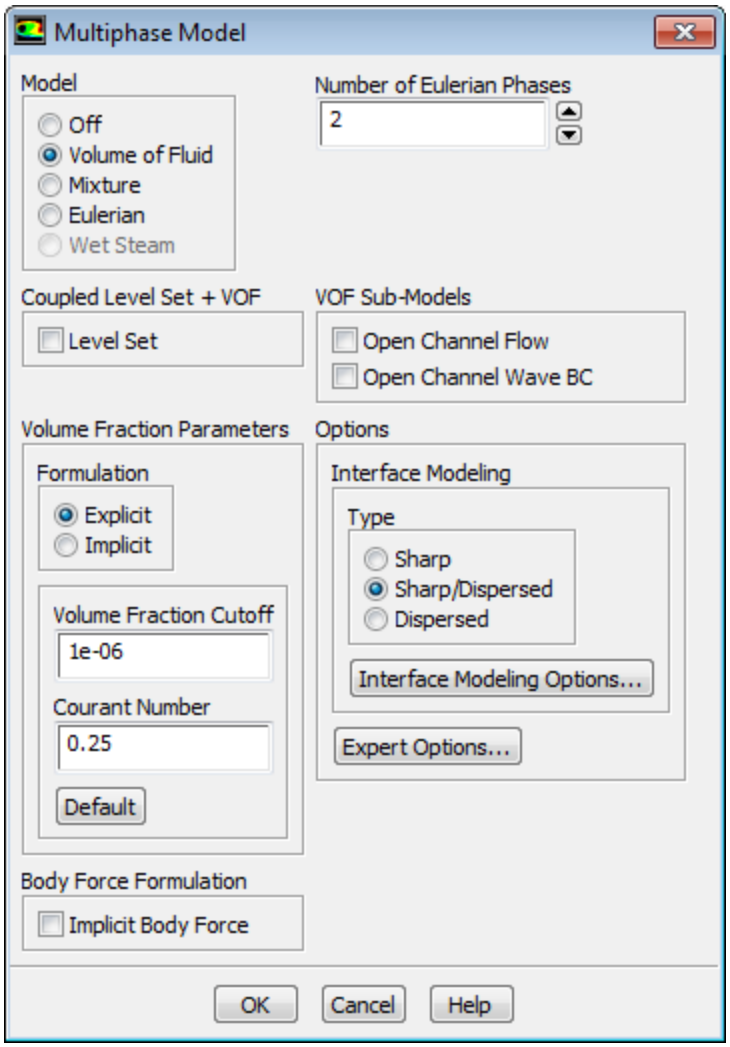
Setup → **Models** → **Multiphase** → **Edit...**

The dialog box will expand to show the relevant inputs for the selected multiphase model.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Volume of Fluid Model Inputs

Figure 24.1: Multiphase Model Dialog Box for the VOF Model

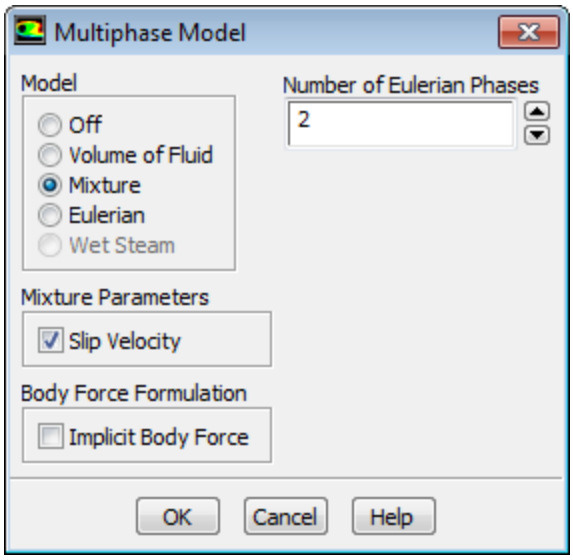


- **Number of Eulerian Phases**
- (optional) **Coupled Level Set + VOF** (see Coupled Level-Set and VOF Model in the Theory Guide)
- volume fraction **Formulation** (**Explicit** or **Implicit**) and associated parameters (see Choosing Volume Fraction Formulation (p. 1371))
- (optional) inclusion of sub-models such as **Open Channel Flow** and **Open Channel Wave BC**
- **Interface Modeling** type and options (see Choosing Volume Fraction Formulation (p. 1371))
- (optional) selection of **Expert Options** (see Expert Options (p. 1374))
- (optional) inclusion of the **Implicit Body Force** formulation (see Including Body Forces (p. 1377))

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mixture Model Inputs

Figure 24.2: Multiphase Model Dialog Box for the Mixture Model

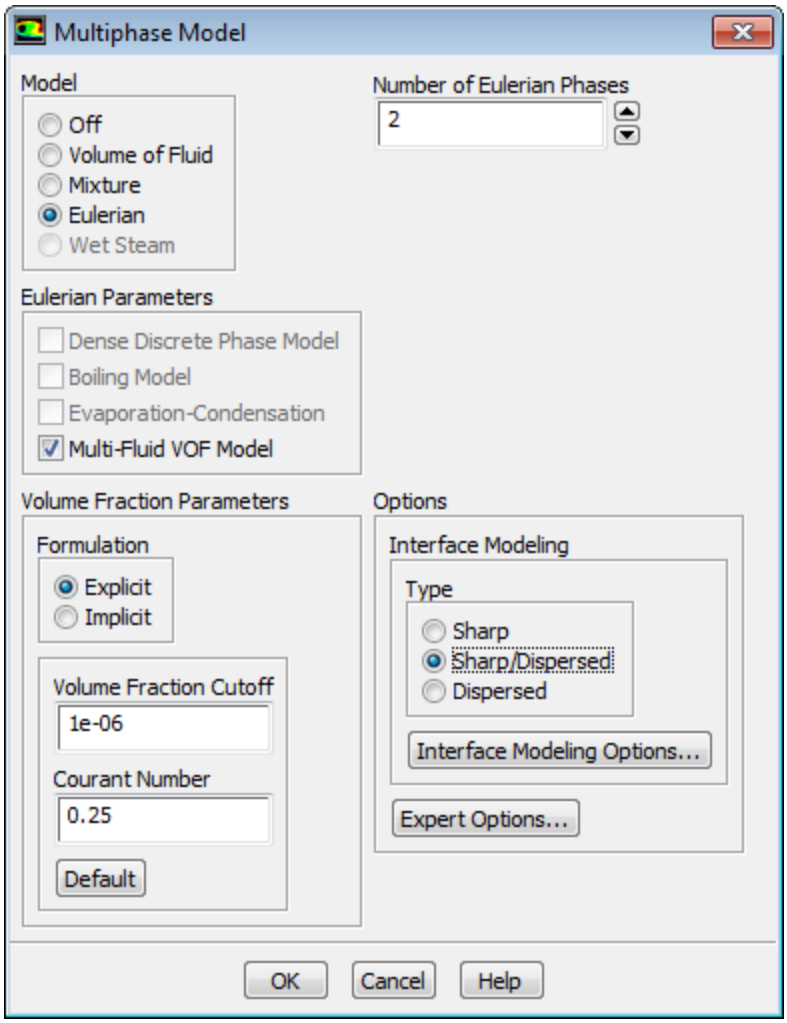


- **Number of Eulerian Phases**
- (optional) inclusion of **Slip Velocity** (see Solving a Homogeneous Multiphase Flow (p. 1375))
- (optional) inclusion of the **Implicit Body Force** formulation (see Including Body Forces (p. 1377))

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Eulerian Model Inputs

Figure 24.3: Multiphase Model Dialog Box for the Eulerian Model



- **Number of Eulerian Phases**
- (optional) inclusion of the **Dense Discrete Phase Model**
- (optional) inclusion of the **Boiling Model**
- (optional) inclusion of **Evaporation-Condensation** modeling
- (optional) inclusion of the **Multi-Fluid VOF Model**
- **Evaporation-Condensation Model Options** where you can choose whether to use the **Lee Model** or **Thermal Phase Change** model for interphase mass transfer.
- volume fraction **Formulation** (**Explicit** or **Implicit**) and associated parameters (see Choosing Volume Fraction Formulation (p. 1371))
- **Interface Modeling** type and options (see Choosing Volume Fraction Formulation (p. 1371)). This is only available if the **Multi-Fluid VOF Model** is enabled.
- (optional) selection of **Expert Options** (see Expert Options (p. 1374))

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To specify the number of phases for the multiphase calculation, enter the appropriate value in the **Number of Eulerian Phases** field. You can specify up to 20 phases.

24.2.2. Choosing Volume Fraction Formulation

ANSYS Fluent supports the following options for volume fraction formulation:

Explicit Formulation
The explicit formulation is non-iterative and is time-dependent. Thus it can only be used with the **Transient** solver. It exhibits better numerical accuracy compared to the implicit formulation. However the time step size is limited by a Courant-based stability criterion.

Implicit Formulation
The implicit formulation is iterative and can be used with either the **Steady** or **Transient** solver. It is well-suited to steady-state applications as the solution information propagates much faster compared to the explicit formulation.

For steady-state flow applications, either a steady-state or a transient simulation may be required, depending on the characteristics of the flow:

- *Steady-state simulation*: If the final steady-state solution is not affected by the initial flow conditions and there is a distinct inflow boundary for each phase.
- *Transient simulation*: If the final steady-state solution is dependant on the initial flow conditions and/or you do not have a distinct inflow boundary for each phase.

For transient flow applications, the implicit formulation may allow you to use a much larger time step than the explicit formulation due to the implicit formulation's unconditional stability. However, the implicit time-step size may be limited by truncation errors. The implicit formulation produces more numerical diffusion than the explicit formulation when using the **First Order** transient formulation. Therefore, the implicit formulation should be used along with higher-order transient formulations to achieve better numerical accuracy.

Both implicit and explicit volume fraction formulations are available for the VOF and Eulerian multiphase models. The mixture multiphase model supports only the implicit formulation.

To specify the volume fraction formulation to be used for the VOF and Eulerian multiphase models, select the appropriate **Formulation** under **Volume Fraction Parameters** in the **Multiphase Model** dialog box.

24.2.2.1. Interface Modeling Type

For the VOF model and for the Eulerian Multiphase model with **Multi-Fluid VOF** enabled, you must specify the interface regime that you will be modeling. This will determine the availability of the various volume fraction discretization schemes (Spatial Discretization Schemes for Volume Fraction (p. 1373)). The following options are available for **Interface Modeling Type**:

Sharp
for when a distinct interface is present between the phases

Dispersed
for when the phases are interpenetrating

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sharp/Dispersed
a hybrid approach for flows consisting of both sharp and dispersed interfaces. This option can also be used to capture mildly sharp interfaces. Mildly sharp interfaces are those that are neither as sharp as would be captured by the schemes available with the **Sharp** option, nor as diffused as would be captured by the schemes available with the **Dispersed** option.

Interfacial Anti-Diffusion

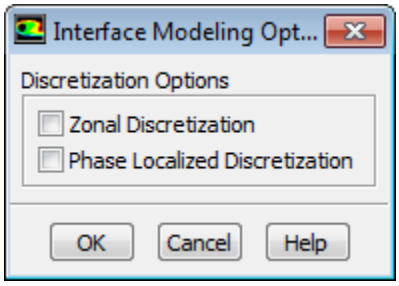
If you are using the Sharp interface modeling type, you can optionally enable the **Interfacial Anti-Diffusion** treatment. This treatment is applied only in interfacial cells and attempts to suppress the numerical diffusion that can arise from the volume fraction advection schemes.

The use of this treatment can have adverse effects on convergence, especially with aggressive settings and large time step size. Therefore, it should be used in cases that use a coarse mesh, that have high aspect-ratio cells, that have large cell-volume jumps in the vicinity of the fluid-fluid interface, or that suffer from excess numerical diffusion.

The strength of the anti-diffusion treatment can be specified using the `define/models/multiphase/interface-modeling-options` text command. The strength can range between `0` (none) and `1` (maximum).

Sharp/Dispersed Interface Modeling Options

Selecting **Sharp/Dispersed** will make available some additional discretization options. To access these, click **Interface Modeling Options...** in the **Multiphase Model** dialog box.



These options allow you to adjust the discretization behavior on a per-zone and/or per-phase-pair basis for cases in which it is important to simultaneously model dispersed and sharp interfaces.

Zonal Discretization
This option is appropriate for applications that require either sharp or dispersed interface modeling depending on the cell zone. After enabling **Zonal Discretization** you will need to specify the slope limiter value on the **Multiphase** tab of the **Fluid** dialog box for each cell zone. (See Defining Multiphase Cell Zone and Boundary Conditions (p. 1391))

Phase Localized Discretization
This option is appropriate for applications that require either sharp or dispersed interface modeling depending on the phase-pair comprising the interface. After enabling **Phase Localized Discretization** you will need to specify the slope limiter values for each phase pair on the **Discretization** tab of the **Phase Interaction** dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.2.2.2. Spatial Discretization Schemes for Volume Fraction

You can choose from various spatial discretization schemes for volume fraction depending on the multiphase model in use, the interface regime type, and the volume fraction formulation chosen. The available discretization schemes for each combination off model, formulation, and interface regime are summarized in the following tables:

Table 24.1: Spatial Discretization Schemes for the VOF and Eulerian with Multi-Fluid VOF Models

Interface Modeling Type	Implicit Formulation	Explicit Formulation
Sharp	Compressive BGM (steady state only) Modified HRIC	Geo-Reconstruct CICSAM Compressive Modified HRIC
Sharp/Dispersed	Compressive Modified HRIC	
Dispersed	First Order Upwind Second Order Upwind QUICK	First Order Upwind QUICK

Table 24.2: Spatial Discretization Schemes for the Eulerian Model without Multi-Fluid VOF

Implicit Formulation	Explicit Formulation
First Order Upwind QUICK Modified HRIC	First Order Upwind QUICK Modified HRIC Compressive

Table 24.3: Spatial Discretization Schemes for the Mixture Model

Implicit Formulation	Explicit Formulation
First Order Upwind QUICK	Not Applicable

The **Geo-Reconstruct** scheme is an interface tracking scheme based on geometrical information. It is the most accurate scheme, but is more computationally expensive than the other schemes. **Geo-Reconstruct** is the preferred scheme when solving on meshes of poor quality.

The **CICSAM**, **Modified HRIC**, and **Compressive** schemes are interface capturing schemes based on algebraic information.

The Donor-Acceptor discretization is a scheme applicable only to quad- and hex-meshes. As it is considered obsolete, it does not appear by default in the graphical user interface. However, it can be made available if required using the following text command:

```
/solve/set/expert
<...>
Allow selection of all applicable discretization schemes? [no] yes
```

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This command can also be used to expose discretization schemes that may be applicable, but are hidden by default according to the tables above.

Important

For the geometric reconstruction and donor-acceptor schemes, if you are using a conformal mesh (that is, if the mesh node locations are identical at the boundaries where two subdomains meet), you must ensure that there are no two-sided (zero-thickness) walls within the domain. If there are, you will need to slit them, as described in Slitting Face Zones (p. 215).

For more details about the implementations of the volume fraction discretization schemes, refer to Interpolation Near the Interface in the *Fluent Theory Guide*.

24.2.2.3. Volume Fraction Limits

The **Volume Fraction Cutoff** allows you to specify a cutoff limit for the volume fraction values. The value that you provide is used as the lower cutoff for the volume fraction. All volume fraction values in the domain below this cutoff value are set to zero. The upper cutoff is calculated as (1.0 - lower cutoff). All volume fraction values above the upper cutoff value are set to 1.0. The default value is 1e-6, which is the recommended value. Using a higher value may lead to a higher volume imbalance.

Important

The **Volume Fraction Cutoff** value can be specified when using the VOF model, or when using the Eulerian Multiphase model with the **Explicit** formulation.

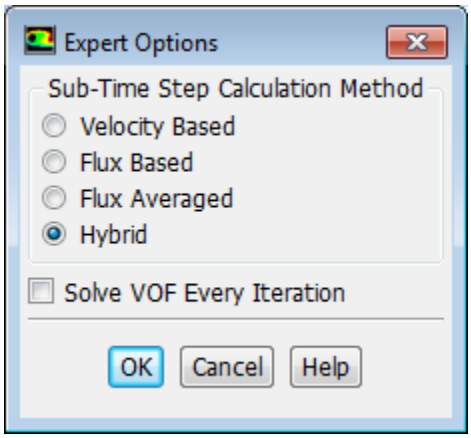
Note

For the **Implicit** formulation, the minimum value allowed is 0 and the maximum allowable value is 1e-6. For the **Explicit** formulation, the minimum value allowed is 0 and the maximum allowable value is 1e-4. A higher cutoff value is available with the **Explicit** formulation because it allows for the local redistribution procedure of partially filled cells, which accounts for volume loss. This treatment is not available with the **Implicit** formulation.

24.2.2.4. Expert Options

When using the **Explicit** formulation, there are several additional options that you can access by clicking **Expert Options...** in the **Multiphase Model** dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Sub-Time Step Calculation Method

This setting controls how the time-scale is computed when determining the time step used for volume fraction.

Solve VOF Every Iteration

This setting controls whether to solve the volume fraction equations once per iteration, or only once per time step.

For a more detailed discussion of how to use these options, refer to Setting Time-Dependent Parameters for the Explicit Volume Fraction Formulation (p. 1445).

24.2.3. Solving a Homogeneous Multiphase Flow

If you are using the mixture model, you have the option to disable the calculation of slip velocities and solve a homogeneous multiphase flow (that is, one in which the phases all move at the same velocity). By default, ANSYS Fluent will compute the slip velocities for the secondary phases, as described in Relative (Slip) Velocity and the Drift Velocity in the Theory Guide. If you want to solve a homogeneous multiphase flow, turn off **Slip Velocity** under **Mixture Parameters**.

24.2.4. The Boussinesq Approximation in Multiphase Flow

When using the Boussinesq approximation for density variation in multiphase flows, there are several important items to consider:

- The Boussinesq approximation is only valid for small variations in fluid densities caused by small variations in temperature. Therefore, constant density is assumed, and a momentum source is only added for modeling buoyancy.
- The Boussinesq approximation assumes that small density variations have little effect on inertial terms and strong effects on gravitational terms. Thus, the differences in specific weights between two fluids or layers of fluids is the force driving the flow. (Waves cannot use this approximation.) Two non-dimensional numbers quantify the effect of buoyancy: the Richardson number and the Rayleigh number.
- Operating density should be set to zero.

For additional information on the Boussinesq approximation, see The Boussinesq Model (p. 833).

24.2.5. Modeling Compressible Flows

If you are using any of the multiphase models for a compressible flow, note the following:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- ANSYS Fluent allows the modeling of:
  - Multiple compressible ideal gas phases and/or incompressible/compressible liquids at any inlet
  - A single real gas or real liquid at any inlet
  - Multiple real gases at a single inlet
  - Multiple compressible liquids using either the Fluent built-in compressible liquid density model based on the Tait law (see Compressible Liquid Density Method (p. 480)) or user-defined functions

  For multiple compressible phases, the fundamental equations for compressible flow remain the same. However, boundary conditions are modified to account for changes in material properties due to compressible fluid mixtures.
- In cases of a mixture of ideal and real gases, the phases must enter the domain through separate inlets.
- When using the VOF model, stability is improved if the primary phase is the compressible ideal gas.
- If you specify the total pressure at a boundary (for example, for a pressure inlet or intake fan), Fluent will use the specified value for temperature at that boundary as total temperature for the compressible phase, and as static temperature for the other phases (which are incompressible).
- For each mass flow inlet, you will need to specify mass flow or mass flux for each individual phase.

Important

Note that if you read a case file that was set up in a version of Fluent previous to 6.1, you will need to redefine the conditions at the mass flow inlets. See Defining Multiphase Cell Zone and Boundary Conditions (p. 1391) for more information on defining conditions for a mass flow inlet in multiphase calculations.

Enhanced Compressible Flow Numerics

Compressible flow simulations may have convergence difficulties due to explicit handling of source term accounting for density variation. These simulations could become extremely unstable in the presence of a large variation in pressure and temperature. In ANSYS Fluent, an enhanced numerical treatment that provides better stability at startup and during calculation of compressible flows is enabled by default. The treatment controls the rate of change of pressure and temperature between iterations and provides additional diagonal dominance to the matrix of equations.

Although disabling the enhanced numerical treatment is not recommended, you can temporarily turn it off to investigate unstable cases using the following text user interface command:

```
solve/set/multiphase-numerics/compressible-flow/enhanced-numerics
...
enhanced compressible flow numerics? [yes] no
```

See Compressible Flows (p. 593) for more information about compressible flows.

Alternative Compressible Flow Numerics

A more general formulation for compressible multiphase flows employs a mathematically rigorous treatment of compressible phases at an inlet boundary. Instead of treating the entering incompressible

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and compressible phases as a pseudo-mixture, this formulation calculates static temperature and pressure using an iterative method based on fundamental thermodynamic relations (such as the Maxwell relations). This approach involves evaluation of the property derivatives, which can be computed either numerically or through analytic expressions. You can enable this alternative treatment via the text command:

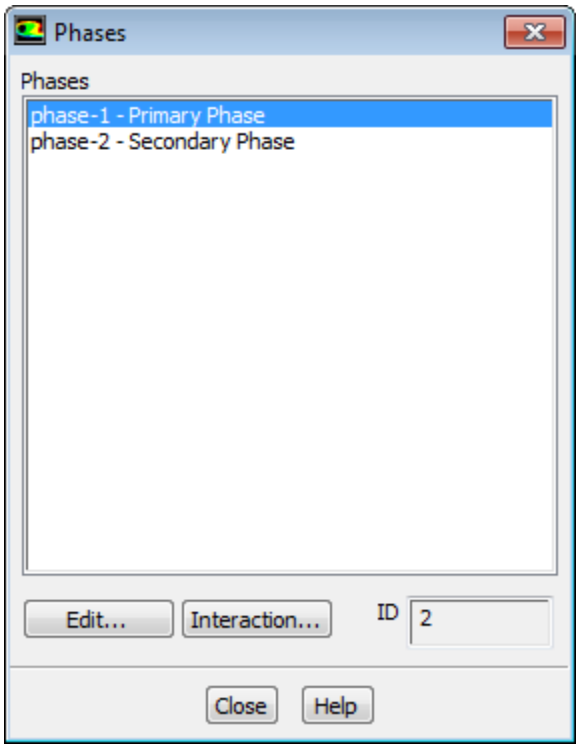
```
solve/set/multiphase-numerics/compressible-flow/alternate-bc-formulation
alternate compressible bc formulation? [no] y
use analytical thermodynamic derivatives? [no] y
```

24.2.6. Defining the Phases

To define the phases (including their material properties) and any interphase interaction (for example, surface tension and wall adhesion for the VOF model, slip velocity for the mixture model, drag functions for the mixture and the Eulerian models), (Figure 24.4: The Phases Dialog Box (p. 1377)).

Setup → Models → Multiphase → Phases Edit...

Figure 24.4: The Phases Dialog Box



Each item in the **Phases** list in this dialog box is one of two types: a **Primary-Phase** indicates that the selected item is the primary phase, and **Secondary-Phase** indicates that the selected item is a secondary phase. To specify any interaction between the phases, click the **Interaction...** button.

Instructions for defining the phases and interaction are provided in Defining the Phases for the VOF Model (p. 1436), Defining the Phases for the Mixture Model (p. 1447), and Defining the Phases for the Eulerian Model (p. 1456) for the VOF, mixture, and Eulerian models, respectively.

24.2.7. Including Body Forces

When large body forces (for example, gravity or surface tension forces) exist in multiphase flows, the body force and pressure gradient terms in the momentum equation are almost in equilibrium while

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the contributions of convective and viscous terms are small in comparison. Segregated algorithms converge poorly unless partial equilibrium of pressure gradient and body forces is taken into account. ANSYS Fluent provides an optional "implicit body force" treatment that can account for this effect, making the solution more robust.

The basic procedure involves augmenting the correction equation for the face flow rate, Equation 21.51 in the Theory Guide, with an additional term involving corrections to the body force. This results in extra body force correction terms in Equation 21.49 in the Theory Guide, and allows the flow to achieve a realistic pressure field very early in the iterative process.

To include this body force, enable **Gravity** in the Operating Conditions Dialog Box (p. 2382) and specify the **Gravitational Acceleration**.

Setup → **Cell Zone Conditions** → **Operating Conditions...**

For VOF calculations, you should also enable the **Specified Operating Density** option in the **Operating Conditions** dialog box, and set the **Operating Density** to be the density of the lightest phase. (This excludes the buildup of hydrostatic pressure within the lightest phase, improving the round-off accuracy for the momentum balance.) If any of the phases is compressible, set the **Operating Density** to zero.

Important

For VOF and mixture calculations involving body forces, it is recommended that you also enable the **Implicit Body Force** treatment for the **Body Force Formulation** in the **Multiphase Model** dialog box. This treatment improves solution convergence by accounting for the partial equilibrium of the pressure gradient and body forces in the momentum equations. See Including Body Forces (p. 1377) for details.

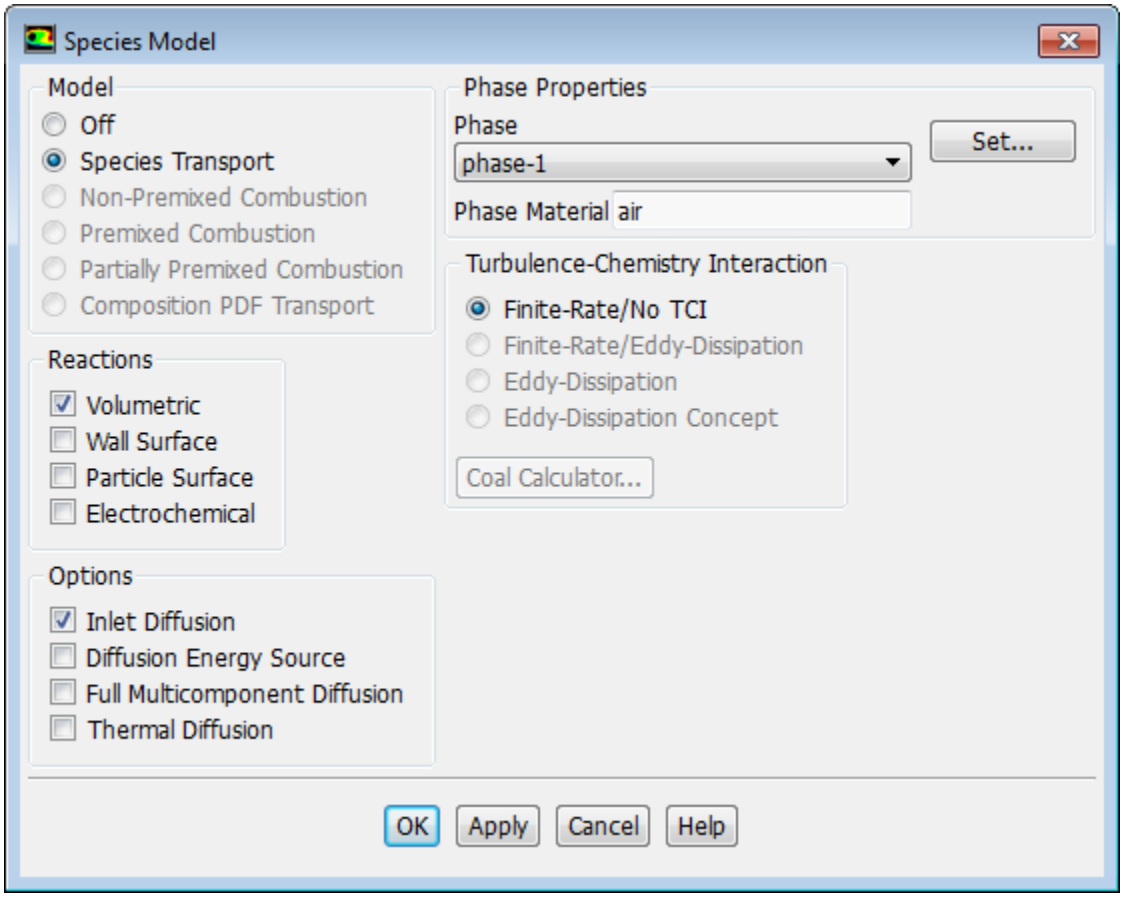
24.2.8. Modeling Multiphase Species Transport

ANSYS Fluent lets you describe a multiphase species transport and volumetric reaction (Modeling Species Transport in Multiphase Flows in the Theory Guide) in a fashion that is similar to setting up a single-phase chemical reaction using the **Species Model** dialog box (for example, Figure 24.5: The Species Model Dialog Box with a Multiphase Model Enabled (p. 1379)).

Setup → **Models** → **Species** → **Edit...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

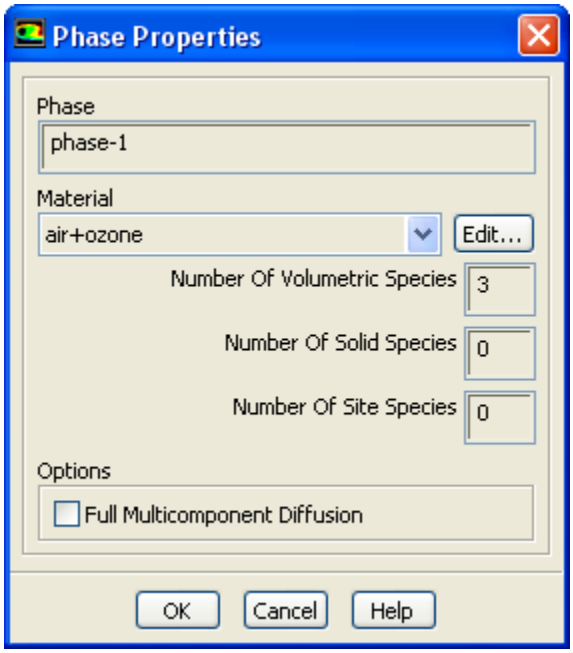
Figure 24.5: The Species Model Dialog Box with a Multiphase Model Enabled



1. Select **Species Transport** under **Model**.
2. Enable **Volumetric** under **Reactions**.
3. Select a specific phase using the **Phase** drop-down list under **Phase Properties**.
4. Click the **Set...** button to display the **Phase Properties** dialog box (Figure 24.6: The Phase Properties Dialog Box (p. 1380)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.6: The Phase Properties Dialog Box



In the **Phase Properties** dialog box, the material for each phase is listed in the **Material** drop-down list. From this list, you can choose the material that you want to use for a specific phase. The drop-down list contains all of the materials, then open the **Edit Material** dialog box by clicking the **Edit...** (or **View...**) button next to the **Material** drop-down list.

5. In the **Species Model** dialog box, choose the **Turbulence-Chemistry Interaction** model. Three models are available:

   Laminar Finite-Rate
   computes only the Arrhenius rate (see Equation 7.7 in the Theory Guide) and neglects turbulence-chemistry interaction.

   Eddy-Dissipation
   (for turbulent flows) computes only the mixing rate (see Equation 7.25 and Equation 7.26 in the Theory Guide).

   Finite-Rate/Eddy-Dissipation
   (for turbulent flows) computes both the Arrhenius rate and the mixing rate and uses the smaller of the two.

When modeling multiphase species transport, additional inputs may also be required depending on your modeling needs. See, for example, Specifying Heterogeneous Reactions (p. 1380) for more information defining heterogeneous reactions, or Including Mass Transfer Effects (p. 1383) for more information on mass transfer effects.

24.2.9. Specifying Heterogeneous Reactions

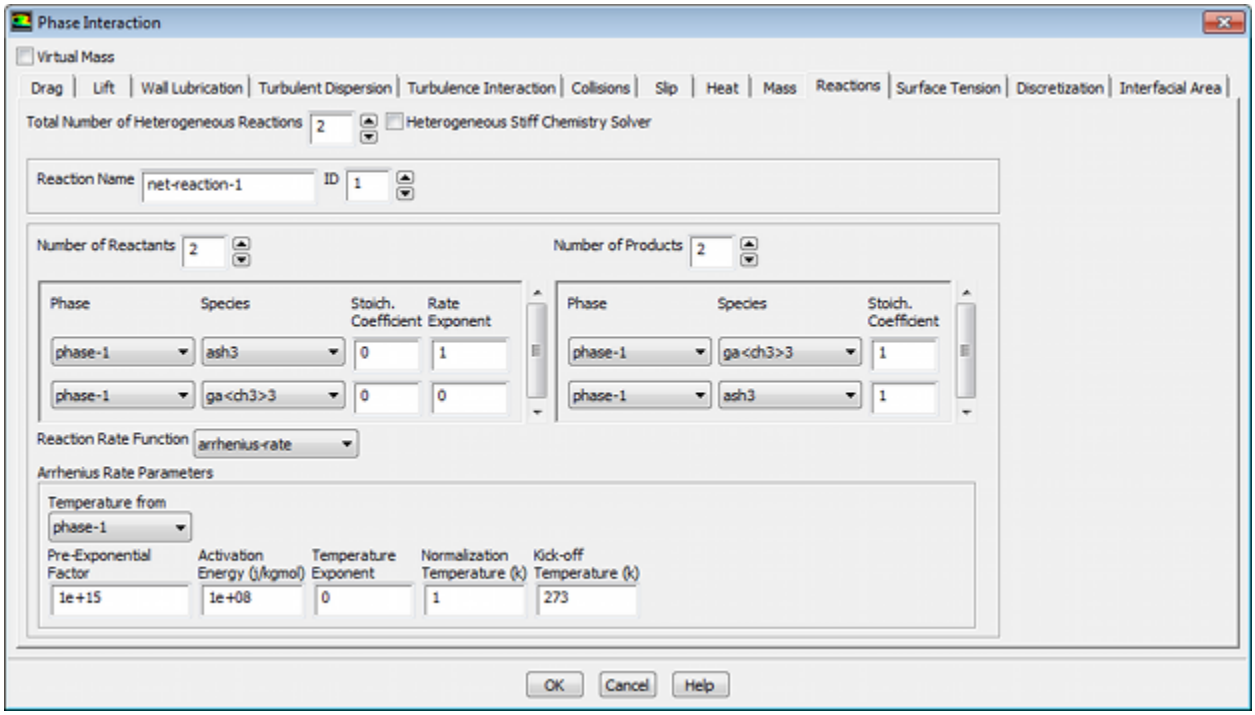
You can use ANSYS Fluent to define multiple heterogeneous reactions and stoichiometry using the **Phase Interaction** dialog box (for example, Figure 24.7: The Phase Interaction Dialog Box for Heterogeneous Reactions (p. 1381)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → Models → Multiphase → Phases → Edit... → Interaction...

1. In the **Phases** dialog box (Figure 24.4: The Phases Dialog Box (p. 1377)), click the **Interaction...** button to open the **Phase Interaction** dialog box.

   Figure 24.7: The Phase Interaction Dialog Box for Heterogeneous Reactions



2. Click the **Reactions** tab in the **Phase Interaction** dialog box.

3. Set the total number of reactions (volumetric reactions, wall surface reactions, and particle surface reactions) in the **Total Number of Heterogeneous Reactions** field. (Use the arrows to change the value, or type in the value and press **Enter**.)

4. Enable the **Heterogeneous Stiff Chemistry Solver** option if your inter-phase reaction mechanism contains numerically stiff reactions. This option can improve convergence and is available for transient Eulerian multiphase simulations. When this option is enabled, ANSYS Fluent uses a fractional step algorithm where the flow is advanced without reaction sources for a time step, and then the chemistry is integrated point-by-point for the same time step. The stiff chemistry scheme solves all species in all phases coupled. Note that it is possible to include homogeneous (intra-phase) reactions along with the heterogeneous reactions in the **(Phase Interaction** dialog box (instead of in the reaction mechanism in the **Create/Edit Materials** dialog box), and these reactions will be solved with the stiff solver. The stiff ODE solver tolerances can be set using the following text command:

   `solve → set → heterogeneous-stiff-chemistry`

5. Specify the **Reaction Name** of each reaction that you want to define.

6. Set the **ID** of each reaction you want to define. (Again, if you type in the value be sure to press **Enter**.)

7. For each reaction, specify how many reactants and products are involved in the reaction by increasing the value of the **Number of Reactants** and the **Number of Products**. Select each reactant or product in the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reaction tab and then set its stoichiometric coefficient in the **Stoich. Coefficient** field. (The stoichiometric coefficient is the constant $\nu'_{i,r}$ or $\nu''_{i,r}$ in Equation 7.6 in the Theory Guide.)

8. For each reaction, indicate the **Phase** and **Species** and the stoichiometric coefficient for each of your reactants and products.

9. For each reaction, use the **Reaction Rate Function** drop-down list to select one of the following:

none
if you do not want to include a reaction rate

population-balance
is the mass transfer due to nucleation and growth. If neither the primary phase or secondary phase has species associated with it, then the mass transfer is modeled as unidirectional. If the mass transfer process involves reactions or species, the problem must be set up as one involving heterogeneous reaction/mass transfer.

Important

The **population-balance** option for **Reaction Rate Function** should not be used if there are multiple reactions leading to the formation of the secondary phase. In this case, the reaction rate functions should be specified either through the standard dialog boxes or user-defined functions and the growth rate function should be a sum of the individual reaction rates. This would ensure consistency between the individual reaction rates and the total mass transfer from the primary to the secondary phase.

You can always use `DEFINE_MASS_TRANSFER` or `DEFINE_HET_RXN_RATE` user-defined function types instead of the **population-balance** option to specify mass transfer rates. However, the growth rate function and the mass transfer rates returned from the UDFs need to be consistent with each other.

arrhenius-rate
to specify rate exponents for an Arrhenius-type reaction (see Heterogeneous Phase Interaction in the Theory Guide for more information)

Important

This simple form of the Arrhenius rate option may only be used for devolatilization reactions only. Char combustion reaction may be more involved and complicated to be simply casted in this form. Additional diffusion rate formulations may be needed to formulate a complete char (or solid phase) reaction system.

Important

Note that you can also specify the heterogeneous reaction rates using a user-defined function. A user-defined function is available for an Arrhenius-type reaction with rate

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

exponents that are equivalent to the stoichiometric coefficients. For more information, see DEFINE_HET_RXN_RATE in the Fluent Customization Manual.

Important

ANSYS Fluent assumes that the reactants are mixed thoroughly *before* reacting together, therefore the heat and momentum transfer is based on this assumption. This assumption can be disabled using a text command. For more information, contact your ANSYS Fluent support engineer.

24.2.10. Including Mass Transfer Effects

As discussed in Modeling Mass Transfer in Multiphase Flows in the Theory Guide, mass transfer effects in the framework of ANSYS Fluent's general multiphase models (that is, Eulerian multiphase, mixture multiphase, or VOF multiphase) can be modeled in one of three ways:

- Unidirectional constant rate mass transfer (not available for VOF calculations)
- UDF-prescribed mass transfer
- mass transfer through cavitation, evaporation-condensation, boiling, or generalized species mass transfer

Because of the different procedures and limitations involved, defining mass transfer through the Singhal et al. cavitation model is described separately in Including Cavitation Effects (p. 1455).

Note

There are limitations and recommendations specific to using cavitation with the VOF model. See Limitations of Cavitation with the VOF Model in the *Fluent Theory Guide* for additional information.

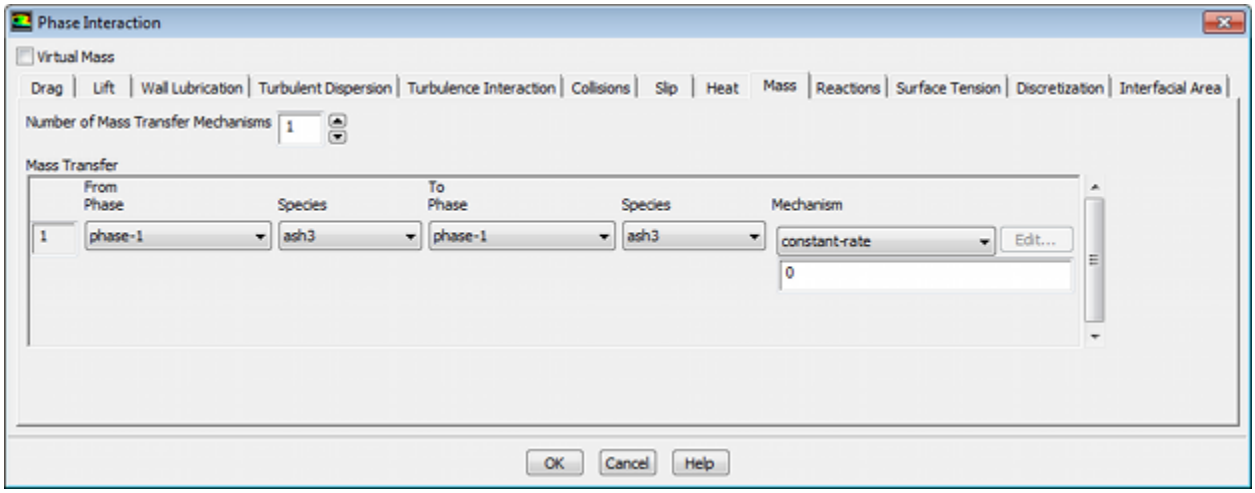
For general limitations regarding the cavitation models, see Limitations of the Cavitation Models in the *Fluent Theory Guide*.

To define mass transfer in a multiphase simulation as a unidirectional constant, using a user-defined function, through population balance, cavitation, evaporation/condensation, or species mass transfer you will need to use the **Phase Interaction** dialog box (for example, Figure 24.8: The Phase Interaction Dialog Box for Mass Transfer (p. 1384)).

Setup → **Models** → **Multiphase** → **Phases** → **Edit...** → **Interaction...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.8: The Phase Interaction Dialog Box for Mass Transfer



1. Click the **Mass** tab in the **Phase Interaction** dialog box.

2. Specify the **Number of Mass Transfer Mechanisms**.

   Important

   In general, you should use only one mass transfer mechanism when you employ one of the available mass transfer models (see Step 7). Although using multiple mass transfer mechanisms is allowed for any pairs of phases (including the same pair of phases), it is not recommended because the mass transfer models have been fundamentally developed for a single mass transfer process. These models do not account for possible interactions between two mass transfer mechanisms in a flow system. In order to account for all relevant physics using UDFs in multiple mass transfer mechanisms, you can use the user-defined option. For additional restrictions when using multiple mass transfer mechanisms, see Step 7.

3. For each mechanism, specify the phase of the source material under **From Phase**. Note that the phase you select for **From Phase** must be a liquid if you plan to select **cavitation**, **evaporation-condensation**, or **boiling** from the **Mechanism** drop-down menu (see Step 7).

4. If species transport is part of the simulation, and the source phase is composed of a mixture material, then specify the species of the source phase mixture material in the corresponding **Species** drop-down list.

5. For each mechanism, specify the phase of the destination material phase under **To Phase**. Note that the phase you select for **To Phase** must be a vapor if you plan to select **cavitation**, **evaporation-condensation**, or **boiling** from the **Mechanism** drop-down menu. If you plan to select **species-mass-transfer** and one of the phases is a gas, it must be specified as the **To Phase** (see Step 7).

6. If species transport is part of the simulation, and the destination phase is composed of a mixture material, then specify the species of the destination phase mixture material in the corresponding **Species** drop-down list.

7. For each pair of phases with mass transfer, select the desired mass transfer mechanism under **Mechanism**. The available mechanisms and their inputs are described in Mass Transfer Mechanisms (p. 1385).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS Fluent will automatically include the terms needed to model mass transfer in all relevant conservation equations. Another option to model mass transfer between phases is through the use of user-defined sources and their inclusion in the relevant conservation equations. This approach is more involved, but more powerful, allowing you to split the source terms according to a model of your choice.

Important

- Except for the **species-mass-transfer** mechanism, if the same mass transfer model is selected for multiple mass transfer mechanisms, the most recently edited mechanism sub-model, including its parameters, options, and constants (excluding its material properties, such as vaporization pressure or saturation temperature), will apply to all the mass transfer mechanisms.
- When modeling multiple **species-mass-transfer** mechanisms, you can select different species mass transfer models and model options, such as Raoult's Law, Henry's Law, etc. However, because there is a lack of interaction between different species mass transfer models, this should be done with caution. For information on modeling restrictions for interphase species mass transfer, see Interphase Species Mass Transfer in the *Fluent Theory Guide*.
- Momentum, energy, and turbulence are also transported with the mass that is transferred. ANSYS Fluent assumes that the reactants are mixed thoroughly *before* reacting together, therefore the heat and momentum transfer is based on this assumption. This assumption can be disabled using a text command. For more information, contact your ANSYS Fluent support engineer.

When your model involves the transport of multiphase species, you can define a mass transfer mechanism between species from different phases. If a particular phase does not have a species associated with it, then the mass transfer throughout the system will be performed by the bulk fluid material.

Important

Including species transport effects in the mass transport of multiphase simulation requires that **Species Transport** be selected in the **Species Model** dialog box.

Setup → **Models** → **Species**

24.2.10.1. Mass Transfer Mechanisms

When including mass transfer effects in a multiphase simulation, you can choose from the following mechanisms for each phase pair that undergoes mass transfer.

constant-rate
enables a user-specified constant, unidirectional mass transfer. The **constant-rate** must be entered in units of 1/time unit.

user-defined
allows you to implement a correlation reflecting a model of your choice, through a user-defined function.

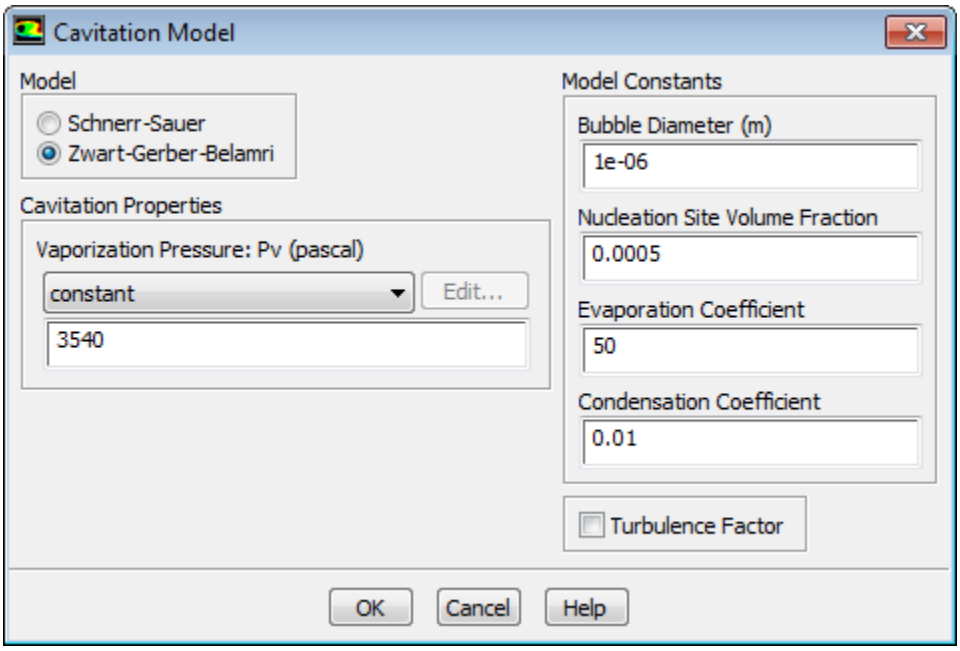
population-balance
allows you to model flow where a number density function is introduced to account for the particle population. With the aid of particle properties (for example, particle size, porosity, composition, and so on),

different particles in the population can be distinguished and their behavior can be described. For a comprehensive understanding of this option, refer to the Population Balance Module Manual.

cavitation

allows you to select a cavitation model. The cavitation models are available when using mixture, VOF, and Eulerian multiphase models. You are provided with two model options: **Schnerr-Sauer** and **Zwart-Gerber-Belamri**. To open the **Cavitation Model** dialog box, select **cavitation** from the **Mechanism** drop-down list. For information about the cavitation models, refer to Cavitation Models in the Theory Guide.

Figure 24.9: The Cavitation Model Dialog Box



Note

If multiple phase pairs have the **cavitation** mechanism selected, the same settings in the **Cavitation Model** dialog box will be used for all phase pairs.

- For the **Schnerr-Sauer** model, specify the **Bubble Number Density** under **Model Constants**.
- For the **Zwart-Gerber-Belamri** model, specify the **Bubble Diameter**, the **Nucleation Site Volume Fraction**, the **Evaporation Coefficient**, and the **Condensation Coefficient** under **Model Constants**.

Note

It is advisable to use the default values for all the model constants in both the **Schnerr-Sauer** and **Zwart-Gerber-Belamri** models.

- Under **Cavitation Properties**, specify the **Vaporization Pressure** and, if desired, a temperature dependence function. In addition to the methods described in Defining Properties Using Temperature-Dependent Functions (p. 475) (**constant**, **polynomial**, **piecewise-linear**, **piecewise-polynomial**, or **user-defined**) you can choose **taylor-approximation**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **taylor-approximation** method models the vaporization pressure as linearly varying with temperature about the free-stream value. This approach can improve numerical stability, but is only appropriate when deviations about the free-stream temperature are sufficiently small that higher-order terms can be neglected. The expression for the vaporization pressure using the **taylor-approximation** method is:

$$P_v = P_v(T_\infty) + C_{thermal} \frac{\rho_l(T_\infty)\rho_v(T_\infty)L(T_\infty)}{T_\infty\left[\rho_l(T_\infty) - \rho_v(T_\infty)\right]}(T - T_\infty) \tag{24.1}$$

where T_∞ is the free-stream temperature, L is the latent heat, and $C_{thermal}$ is a user-modifiable coefficient. In the **Taylor Approximation** dialog box, you will specify **Vaporization Pressure** (at the free-stream temperature), **Free Stream Temperature**, and **Thermal Coefficient** ($C_{thermal}$).

- If you want to include the influence of turbulence on the threshold cavitation pressure, enable **Turbulence Factor**. If desired, you can adjust the value of the **Turbulent Coefficient**. For details about the implementation of the Turbulence Factor, see Turbulence Factor in the *Fluent Theory Guide*.

Note

If the **Mixture** multiphase model is enabled, then the Singhal et al. cavitation model can be enabled using the `solve/set/expert` text command and responding `yes` to `use Singhal-et-al cavitation model?`. The **Singhal-Et-Al Cavitation Model** option will now be visible in the **Phase Interaction** dialog box, under the **Mass** tab. Enable this option to include the Singhal et al. cavitation model. Refer to Including Cavitation Effects (p. 1455) for information about setting the cavitation parameters. Also refer to Cavitation Models in the Theory Guide for information about the Singhal et al. model.

To disable this model, first deselect the **Singhal-Et-Al Cavitation Model** option in the **Phase Interaction** dialog box, then type the `solve/set/expert` text command again and enter `no` when asked if you want to `use Singhal-et-al cavitation model?`

---

evaporation-condensation

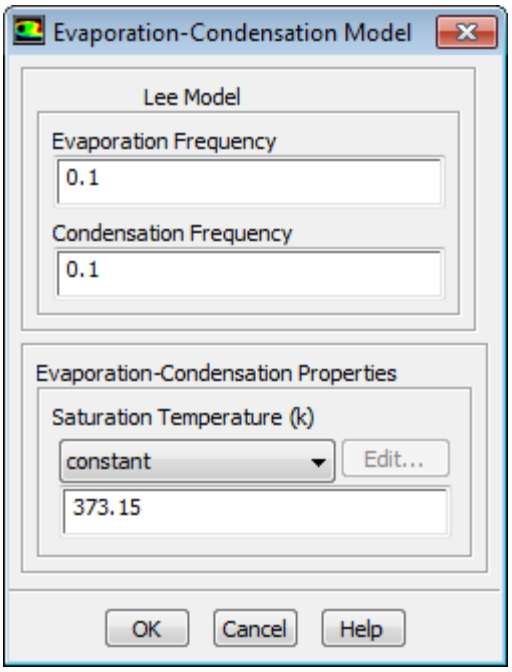
enables you to apply the evaporation-condensation model as the mass transfer mechanism. This model is available with the mixture, VOF, and Eulerian multiphase models. Refer to Evaporation-Condensation Model in the Theory Guide for a theoretical discussion about this model.

Note

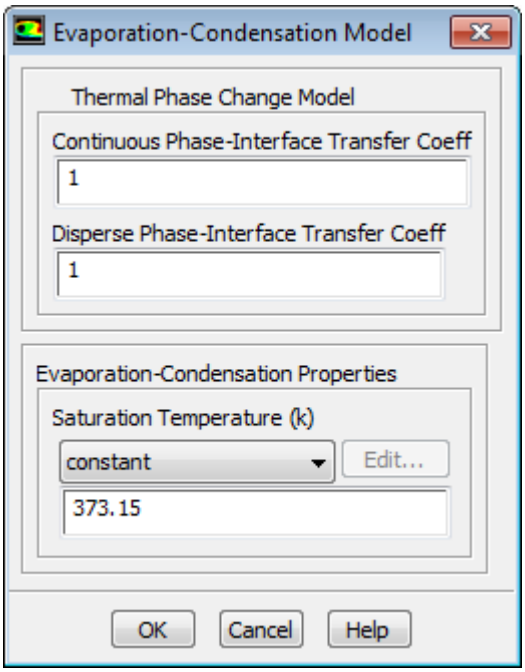
- If multiple phase pairs have the **evaporation-condensation** mechanism selected, the same settings in the **Evaporation-Condensation Model** dialog box will be used for all phase pairs.
- If you are using the **evaporation-condensation** mass transfer mechanism with VOF it is recommended that you use one of the diffusive interface capturing discretization schemes for volume fraction, such as **QUICK**, **HRIC**, or **Phase Localized Compressive** (Discretizing Using the Phase Localized Compressive Scheme (p. 1443)).

- If you are using the mixture or VOF multiphase formulation, or if you have selected the **Lee Model** in the Eulerian formulation, you will need to enter the **Evaporation Frequency** and **Condensation Frequency** model constants. These values correspond to the coefficient $coeff$ (Equation 17.513 in the Theory Guide). The values are `0.1` by default. However, note that the bubble diameter and accommodation coefficient are usually not very well known, so the appropriate values for a given problem can be very different. It is important that the values be tuned to match experimental data.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- If you are using the Eulerian multiphase formulation and have selected the **Thermal Phase Change** model, enter the **Continuous Phase–Interface Transfer Coeff** and **Dispersed Phase–Interface Transfer Coeff**. If you are using the two-resistance option for heat transfer, these values act as multipliers for the phase heat transfer coefficients determined for each phase and the default value of 1 is usually appropriate. If you are using one of the other heat transfer options, then these parameters correspond to the coefficient $coeff$ (Equation 17.513 in the Theory Guide) and should be tuned based on experimental data.



- Specify the **Saturation Temperature** for your flow regime.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

species-mass-transfer

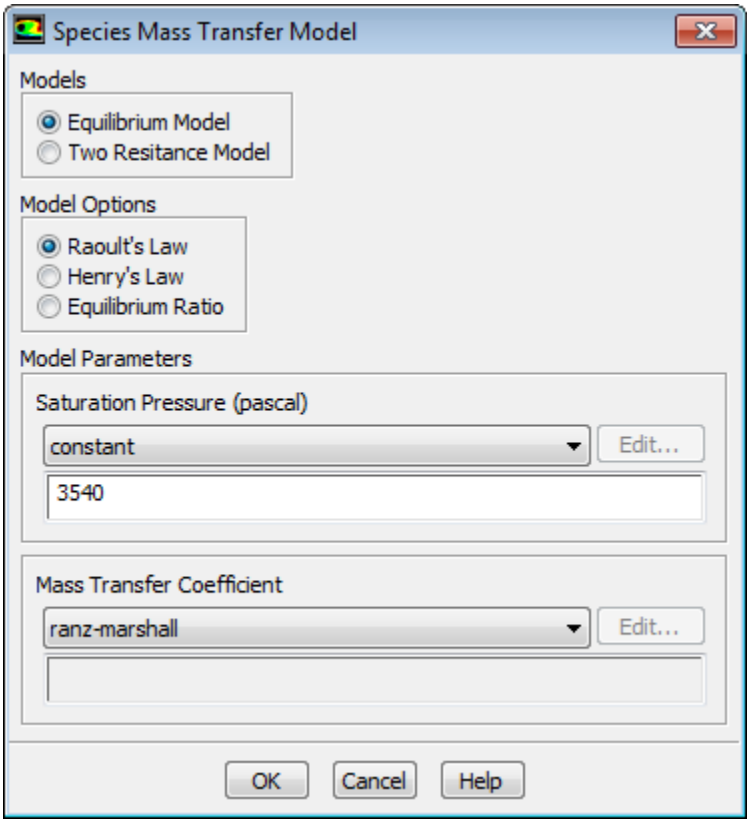
enables you to model generalized interphase species mass transfer in either the Mixture model (when Slip Velocity is enabled) or the Eulerian model subject to the following conditions:

- Both phases consist of mixtures with at least two species, and at least one of species is present in both phases.
- The two mixture phases are in contact and separated by an interface.
- Species mass transfer can only occur between the same species from one phase to the other. For example, evaporation/condensation between water liquid and water vapor.
- As in all interphase mass transfer models in Fluent, if a gas mixture is involved in an interphase species mass transfer process, it is always treated as a **To Phase** and the liquid mixture as a **From Phase**.
- For the species involved in the mass transfer, the mass fractions in both phases must be determined by solving transport equations. For example, the mass fractions of water liquid and water vapor in an evaporation/condensation case must be solved directly from the governing equations, rather than algebraically or from the physical constraint relations.

When you select **species-mass-transfer** under **Mechanism** in the Phase Interactions dialog box, you will be prompted with the **Species Mass Transfer Model** dialog box.

Note

If multiple phase pairs have the **species-mass-transfer** mechanism selected, the same settings in the **Species Mass Transfer Model** dialog box will be used for all phase pairs.



The steps to configure the species mass transfer model are as follows:

1. Under **Models** select either the **Equilibrium Model** or the **Two-Resistance Model**.

   Equilibrium Model
   models the species mass transfer based on equilibrium between the bulk phases.

   Two-Resistance Model
   models the species mass transfer based on equilibrium at the phase interface.

2. Under **Model Options**, choose the sub-model for determining the dynamic equilibrium relations between the same species in a pair of phases.

   Raoult's Law
   Use Raoult's Law as described in Raoult's Law in the *Fluent Theory Guide*. This model is appropriate for gas-liquid cases in which the liquid phase can be modeled as an ideal liquid mixture. If you use Raoult's Law, you must specify the saturation pressure as a constant, polynomial, piecewise-linear, piece-wise polynomial, or as a user-defined function using the `DEFINE_PROPERTY` macro.

   Henry's Law
   Use Henry's Law as described in Henry's Law in the *Fluent Theory Guide*. This model is appropriate for cases in which a gas species is dissolved into a non-ideal liquid mixture phase. If you use Henry's Law you must specify the method for the **Molar Fraction Henry Coeff** or **Molar Concentration Henry Coeff**. You can choose a constant, **vant-hoff** (to use the Van't Hoff correlation), or a user-defined function using the `DEFINE_MASS_TRANSFER` macro. For the **vant-hoff** model, you will need to specify **Reference Henry Constant** (K_H^0 in Equation 17.540 in the *Fluent Theory Guide*) and **Temperature Dependence** ($\frac{\Delta_{soln}H}{R}$) in the **Vant Hoff's Correlation** dialog box.

   Equilibrium Ratio
   Use the Equilibrium Ratio method as described in Equilibrium Ratio in the *Fluent Theory Guide*. This model allows you to specify the equilibrium ratio as a constant, or as a user-defined function. If you use **Equilibrium Ratio** you must specify a constant or user-defined function using the `DEFINE_MASS_TRANSFER` macro for **Molar Fraction Equilibrium Ratio**, **Molar Concentration Equilibrium Ratio**, or **Mass Fraction Equilibrium Ratio**.

3. Choose the sub-model for determining the **Mass Transfer Coefficient**. If you are using the Two Resistance Model, you will need to specify sub-models for both **Continuous-Interface Mass Transfer Coeff.** and **Dispersed-Interface Mass Transfer Coeff.**.

   constant
   Specify a constant value for the mass transfer coefficient.

   sherwood-number
   Specify a constant value for the Sherwood Number. The mass transfer coefficient is calculated from Equation 17.543 in the *Fluent Theory Guide*.

   ranz-marshall
   Use the Ranz-Marshall model as described in Ranz-Marshall Model in the *Fluent Theory Guide*. The Ranz-Marshall model is based on boundary layer theory for steady flow past a spherical particle. It is applicable under the following flow conditions:

$$0 \le \mathrm{Re}_q < 200$$
$$0 \le \mathrm{Sc}_q < 250$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

hughmark
Use the Hughmark model as described in Hughmark Model in the *Fluent Theory Guide*. The Hughmark model extends the Ranz-Marshall model for a wider range of relative Reynolds number, Re_q.

zero-resistance
Specify zero-resistance mass transfer between the bulk phase and interface (available only for the Two-Resistance model).

user-defined
Use a user-defined function for the mass transfer coefficient defined with the `DEFINE_MASS_TRANSFER` macro.

When using the Two-Resistance model, the selection of physically correct mass transfer correlations is highly problem dependent. In the case of mass transfer between a continuous phase and a dispersed phase of approximately spherical particles, the following should be adequate under most situations:

- **ranz-marshall** or **hughmark** for the continuous phase
- **zero-resistance** for the dispersed phase

Using zero-resistance for the dispersed phase implies that the mass transfer occurs very quickly from the interface to the bulk of the dispersed phase. Consequently the interface concentration and bulk concentration in the dispersed phase are equal. This is usually valid for droplet evaporation or gas absorption/dissolution in bubbles. If there exists significant resistance to the mass transfer on the dispersed phase side, one of the other correlations should be chosen for the dispersed phase.

boiling
enables you to apply the boiling model as the mass transfer mechanism. This model is only available with the Eulerian multiphase model (when **Boiling** is enabled in the **Multiphase Model** dialog box). For information about the inputs in Figure 24.62: The Boiling Model Dialog Box (p. 1490), refer to Including the Boiling Model (p. 1486).

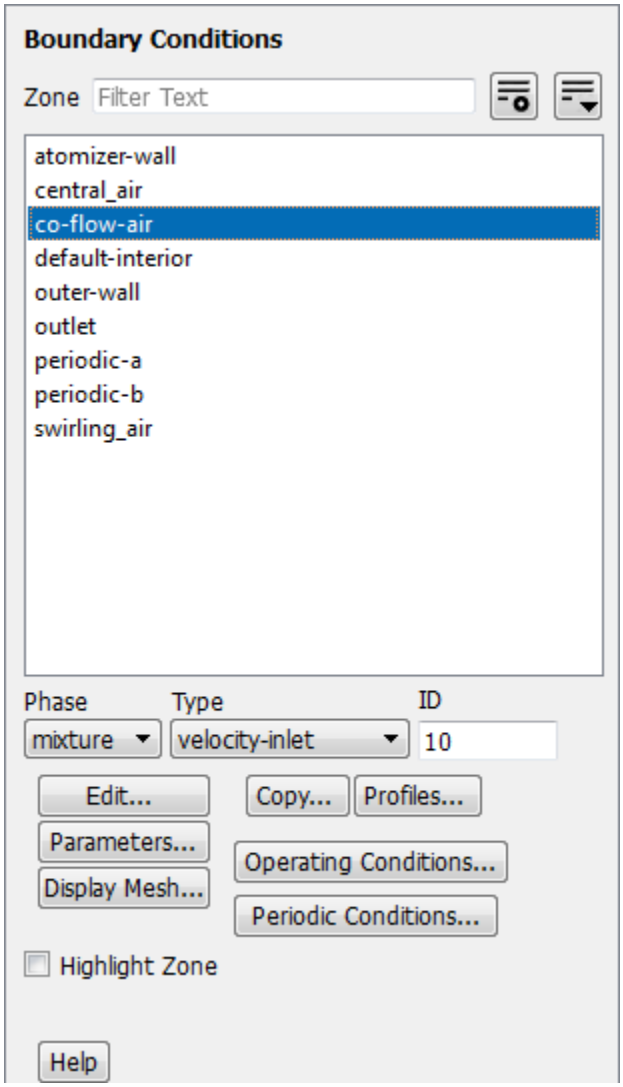
24.2.11. Defining Multiphase Cell Zone and Boundary Conditions

The procedure for setting multiphase cell and boundary conditions is slightly different than for single-phase models. You will need to set some conditions separately for individual phases, while other conditions are shared by all phases (that is, the mixture), as described in Boundary and Cell Zone Conditions for the Mixture and the Individual Phases (p. 1398).

24.2.11.1. Steps for Setting Boundary Conditions

Setup → Boundary Conditions

Figure 24.10: The Boundary Conditions Task Page



The steps you need to perform for each boundary are as follows:

1. Select the boundary in the **Zone** list in the **Boundary Conditions** task page.

2. Set the conditions for the mixture at this boundary, if necessary. (For information about which conditions need to be set for the mixture, refer to Boundary and Cell Zone Conditions for the Mixture and the Individual Phases (p. 1398).)

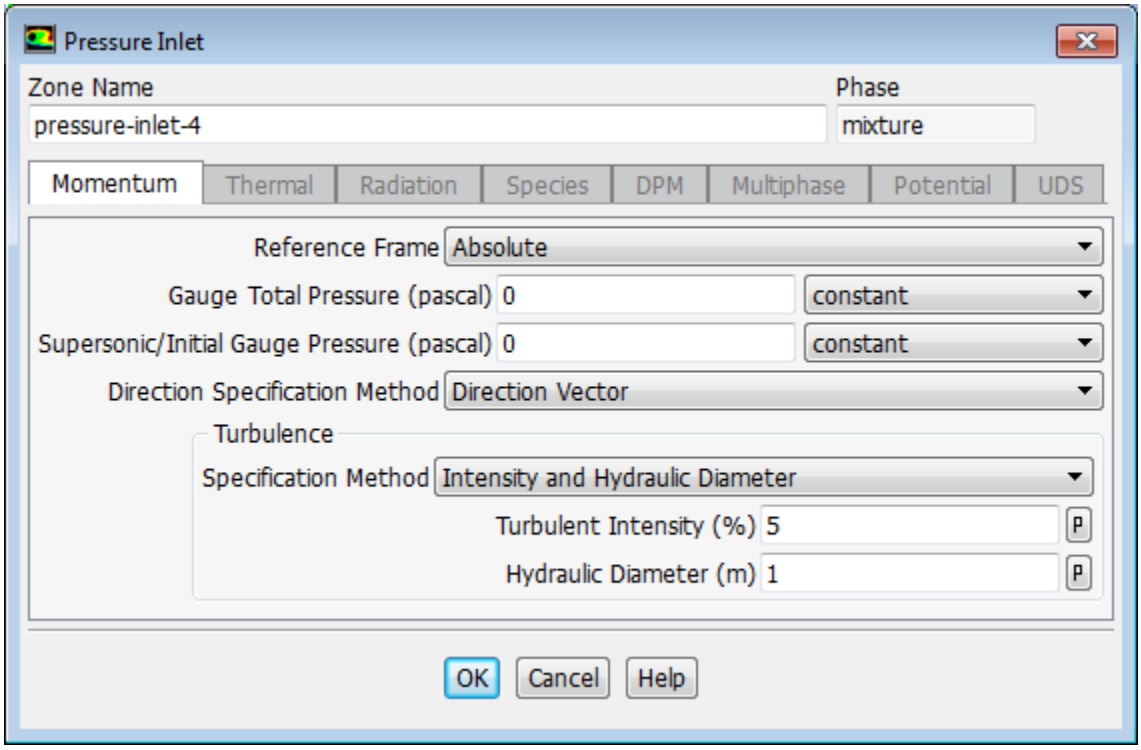
   a. In the **Phase** drop-down list, select **mixture**.

   b. If the current **Type** for this zone is correct, click **Edit...** to open the corresponding boundary condition dialog box (for example, the **Pressure Inlet** dialog box); otherwise, choose the correct zone type in the **Type** drop-down list, confirm the change (when prompted), and the corresponding dialog box will open automatically.

   c. In the boundary condition dialog box for the selected zone type (for example, the **Pressure Inlet** dialog box for the Eulerian model, shown in Figure 24.11: The Pressure Inlet Dialog Box for a Mixture (p. 1393)), specify the mixture boundary conditions.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.11: The Pressure Inlet Dialog Box for a Mixture



Note that only those conditions that apply to all phases, as described in Boundary and Cell Zone Conditions for the Mixture and the Individual Phases (p. 1398), will appear in this dialog box.

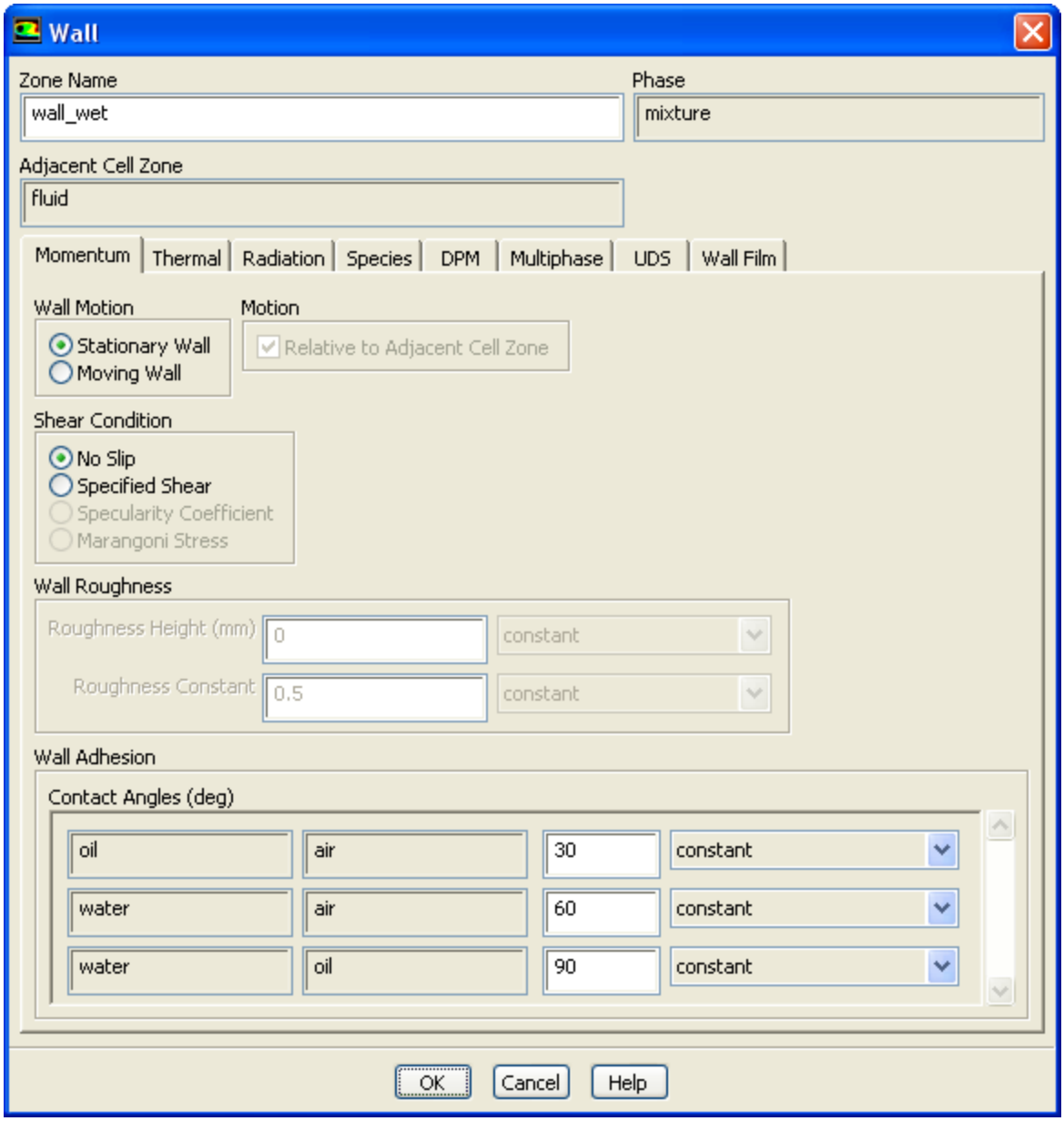
For information about inputs related to the relevant boundary conditions, refer to the appropriate sections for those boundary conditions.

d. Define additional boundary conditions that are specific to the multiphase models.

   i. (VOF and Eulerian models) If you enabled the **Wall Adhesion** option in the **Phase Interaction** dialog box, you can specify the contact angle at the wall for each pair of phases as a constant (as shown in Figure 24.12: The Wall Dialog Box for a Mixture in a VOF or Eulerian Multiphase Calculation with Wall Adhesion (p. 1394)) or a user-defined function (see the Fluent Customization Manual for more information).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.12: The Wall Dialog Box for a Mixture in a VOF or Eulerian Multiphase Calculation with Wall Adhesion



The contact angle (θ_w in Figure 24.31: Measuring the Contact Angle (p. 1442)) is the angle between the wall and the tangent to the interface at the wall, measured inside the phase listed in the left column under **Wall Adhesion** in the **Momentum** tab of the **Wall** dialog box. For example, if you are setting the contact angle between the oil and air phases in the **Wall** dialog box shown in Figure 24.12: The Wall Dialog Box for a Mixture in a VOF or Eulerian Multiphase Calculation with Wall Adhesion (p. 1394), θ_w is measured inside the oil phase.

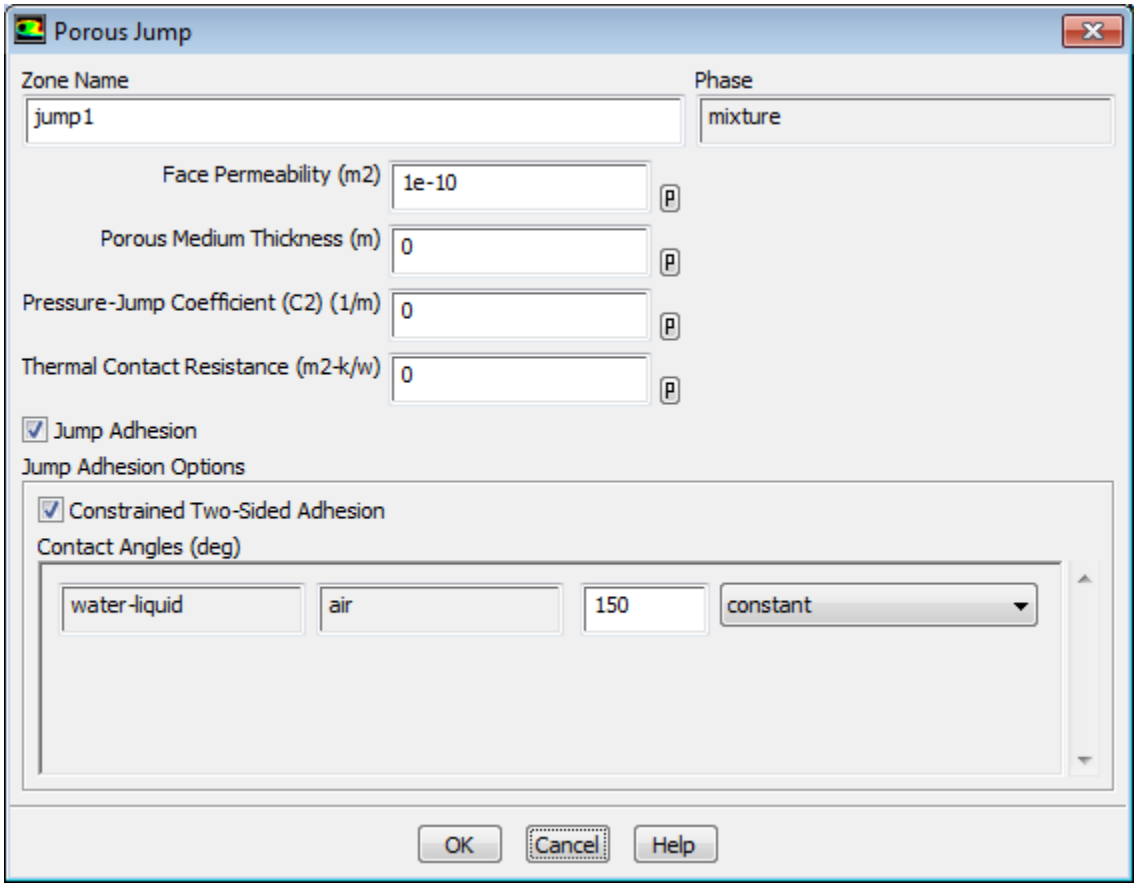
The default value for all pairs is 90 degrees, which is equivalent to no wall adhesion effects (that is, the interface is normal to the adjacent wall). A contact angle of 45°, for example, corresponds to water creeping up the side of a container, as is common with water in a glass.

ii. (VOF model) If you enabled the **Jump Adhesion** option in the **Phase Interaction** dialog box, specify the contact angle at the porous jump for each pair of phases. If you enable the **Jump Adhesion** option in the **Phase Interaction** dialog box, this option becomes visible at each of the porous jump

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

boundaries. You can enable or disable the **Jump Adhesion** option in the **Porous Jump** dialog boxes and provide the inputs for the contact angle at the desired porous jump boundary, as shown in Figure 24.13: The Porous Jump Dialog Box Displaying Jump Adhesion (p. 1395).

Figure 24.13: The Porous Jump Dialog Box Displaying Jump Adhesion



The contact angle θ_w is the angle at the porous jump. To constrain the contact angle at the porous jump based on porous or non-porous fluid zones, enable the **Constrained Two-Sided Adhesion** option. Otherwise, if it is disabled, then the forced two-sided adhesion treatment is in effect. For more detail, see Jump Adhesion.

e. Click **OK** once you complete the boundary condition definition for the mixture.

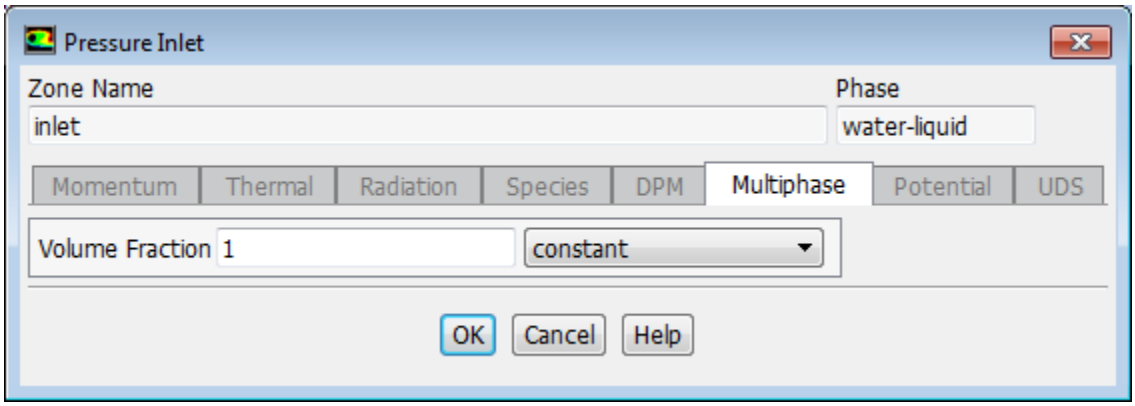
3. Set the conditions for each phase at this boundary, if necessary. (See Boundary and Cell Zone Conditions for the Mixture and the Individual Phases (p. 1398) for information about which conditions need to be set for the individual phases.)

   a. In the **Phase** drop-down list, select the phase (for example, **water**).

> **Important**
>
> Note that, when you select one of the individual phases (rather than the mixture), only one type of zone appears in the **Type** drop-down list. It is not possible to assign phase-specific zone types at a given boundary; the zone type is specified for the mixture, and it applies to all of the individual phases.

b. Click **Edit...** to open the boundary condition dialog box for this phase (for example, the **Pressure Inlet** dialog box, shown in Figure 24.14: The Pressure Inlet Dialog Box for a Phase (p. 1396)).

Figure 24.14: The Pressure Inlet Dialog Box for a Phase

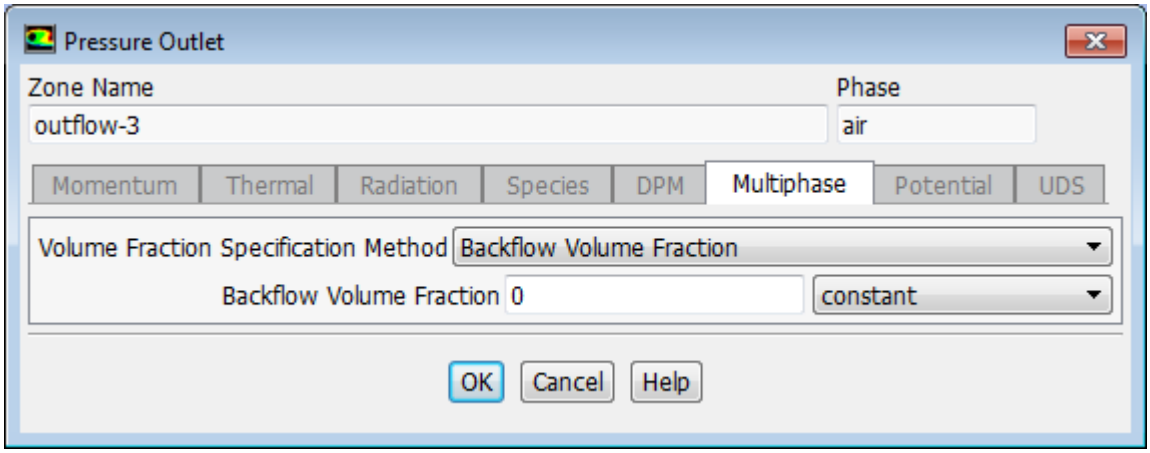


c. Specify the boundary conditions for the phase. Note that only those conditions that apply to the individual phase, as described in Boundary and Cell Zone Conditions for the Mixture and the Individual Phases (p. 1398), will appear in this dialog box.

d. (Secondary phase only) For the pressure outlet, exhaust fan or outlet vent boundary, you can specify the volume fraction specification method as one of the following options:

- **Backflow Volume Fraction**: You can define the backflow volume fraction as a constant, a profile (see Profiles (p. 435)), or a user-defined function (see the Fluent Customization Manual).

Figure 24.15: The Pressure Outlet Dialog Box for a Phase



- **From Neighboring Cell**: This option does not require the specification of the backflow volume fraction, and the volume fraction value is directly taken from the neighboring cells.

  In the case of reverse flow at the pressure outlet boundary, sometimes it becomes tricky to specify the correct values for the backflow volume fraction. Incorrect specification might lead to instability, convergence issues and unphysical behaviour. The backflow volume fraction specification from the neighboring interior cells can be a good choice if information at the pressure outlet boundary is not known.

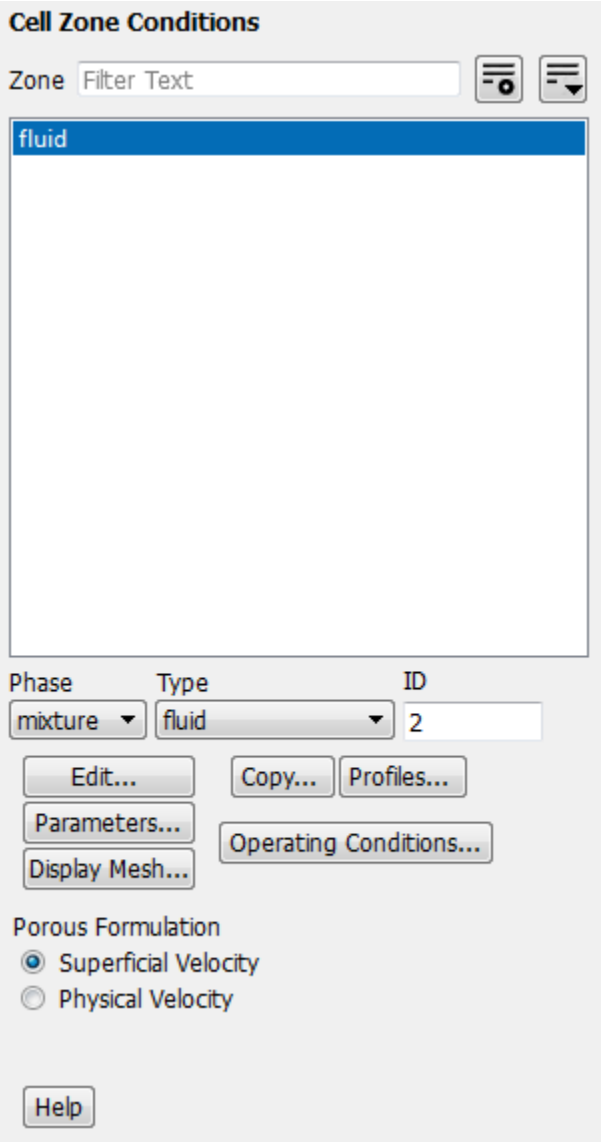
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

e. Click **OK** once you complete the boundary condition definition for the phase.

24.2.11.2. Steps for Setting Cell Zone Conditions

Setup → Cell Zone Conditions

Figure 24.16: The Cell Zone Conditions Task Page



The steps for specifying the mixture and individual phase cell zone conditions are listed below:

1. Select the cell zone in the **Zone** list in the **Cell Zone Conditions** task page.

2. If needed, set the cell zone conditions for the mixture. (For information about which conditions need to be set for the mixture, refer to Boundary and Cell Zone Conditions for the Mixture and the Individual Phases (p. 1398).)

   a. In the **Phase** drop-down list, select **mixture**.

   b. Make sure that the selected **Type** is appropriate for this zone and click **Edit...**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. In the **Fluid** dialog box, set the cell zone conditions that are applicable to the mixture.

d. Define additional cell zone conditions that are specific to the multiphase models.

i. (VOF and Eulerian models) If **Zonal Discretization** is enabled in the **Interface Modeling Options** dialog box, you can specify the **Compressive Scheme Slope Limiter** in the **Multiphase** tab of the **Fluid** dialog box. The setting of the slope limiter can be any value between 0 and 2, inclusive. Values of 0, 1, and 2 correspond to first order upwind, second order upwind, and compressive discretization, respectively with intermediate values resulting in a blended formulation. For more details about the slope limiter, refer to The Compressive Scheme and Interface-Model-based Variants in the *Fluent Theory Guide*.

3. If required, set the conditions for each phase. (See Boundary and Cell Zone Conditions for the Mixture and the Individual Phases (p. 1398) for information about which conditions need to be set for the individual phases.)

a. In the **Phase** drop-down list, select the phase (for example, **water**) and click **Edit...**.

b. In the **Fluid** dialog box, set the cell zone conditions that are applicable to the individual phase.

For more information about inputs for cell zone conditions, see Fluid Conditions (p. 252).

24.2.11.3. Boundary and Cell Zone Conditions for the Mixture and the Individual Phases

The boundary conditions you need to specify for the mixture and for the individual phases will depend on which multiphase model you are using.

Details for each model are provided in the following sections:

24.2.11.3.1. VOF Model
24.2.11.3.2. Mixture Model
24.2.11.3.3. Eulerian Model

24.2.11.3.1. VOF Model

If you are using the VOF model, the conditions you need to specify for each type of boundary zone are listed below and summarized in Table 24.4: Phase-Specific and Mixture Conditions for the VOF Model (p. 1399).

- For an inlet vent, intake fan, pressure inlet, or velocity inlet, there are no conditions to be specified for the primary phase. For each secondary phase, you will need to set the volume fraction as a constant, a profile (see Profiles (p. 435)), or a user-defined function (see the Fluent Customization Manual). All other conditions are specified for the mixture.

- For an exhaust fan, outlet vent, or pressure outlet, there are no conditions to be specified for the primary phase. For each secondary phase, you will need to specify the volume fraction specification method as either **Backflow Volume Fraction** or **From Neighboring Cell** (see step 3. in Steps for Setting Boundary Conditions (p. 1391)). All other conditions are specified for the mixture.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For a mass flow inlet, you will need to set the mass flow rate, mass flux, or average mass flux for each individual phase. All other conditions are specified for the mixture.

  Important

  Note that if you read a VOF case that was set up in a version of ANSYS Fluent prior to 6.1, you will need to redefine the conditions at the mass flow inlets.

- For an axis, fan, outflow, periodic, porous jump, radiator, solid, symmetry, or wall zone, all conditions are specified for the mixture. There are no conditions to be set for the individual phases.
- For a wall zone, you can specify the contact angle for the mixture if the wall adhesion option is enabled.
- For a fluid zone, mass sources are specified for the individual phases, and all other sources are specified for the mixture.
  - If the fluid zone is not porous, all other conditions are specified for the mixture.
  - If the fluid zone is porous, you will enable the **Porous Zone** option in the **Fluid** dialog box for the mixture. The porosity inputs (if relevant) are also specified for the mixture. The resistance coefficients and direction vectors, however, are specified separately for each phase. See User Inputs for Porous Media (p. 267) for the mixture.
  - If **Open Channel Flow** and/or **Open Channel Wave BC** is/are enabled in the **Multiphase Model** dialog box, then the **Numerical Beach** option becomes available under the **Multiphase** tab of the **Fluid** dialog box. To learn how to include numerical beach in your simulation, refer to Numerical Beach Treatment for Open Channels (p. 1432).

See Cell Zone and Boundary Conditions (p. 237) for details about the relevant conditions for each type of boundary. Note that the pressure far-field boundary is not available with the VOF model.

Table 24.4: Phase-Specific and Mixture Conditions for the VOF Model

Type	Primary Phase	Secondary Phase	Mixture
inlet vent, intake fan, pressure inlet, velocity inlet	nothing	volume fraction	all others
exhaust fan, outlet vent, pressure outlet	nothing	volume fraction specification method	all others
mass flow inlet	mass flow/flux	mass flow/flux	all others
axis; fan; outflow; periodic; porous jump; radiator; solid; symmetry; wall	nothing	nothing	all others
pressure far-field	not available	not available	not available
fluid	mass source; other porous inputs	mass source; other porous inputs	porous zone; porosity; all others

24.2.11.3.2. Mixture Model

If you are using the mixture model, the conditions you need to specify for each type of boundary zone are listed below and summarized in Table 24.5: Phase-Specific and Mixture Conditions for the Mixture Model (p. 1400).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For an exhaust fan, outlet vent, or pressure outlet, there are no conditions to be specified for the primary phase. For each secondary phase, you will need to specify the volume fraction specification method as either **Backflow Volume Fraction** or **From Neighboring Cell** (step 3 in Steps for Setting Boundary Conditions (p. 1391). All other conditions are specified for the mixture.

- For an inlet vent, intake fan, or pressure inlet, you will specify for the mixture which direction specification method will be used at this boundary (**Normal to Boundary** or **Direction Vector**). If you select the **Direction Vector** specification method, you will specify the coordinate system (3D only) and flow-direction components for the individual phases. For each secondary phase, you will need to set the volume fraction (as described above). All other conditions are specified for the mixture.

- For a mass flow inlet, you will need to set the mass flow rate, mass flux, or average mass flux for each individual phase. All other conditions are specified for the mixture.

  Important

  Note that if you read a mixture multiphase case that was set up in a version of ANSYS Fluent previous to 6.1, you will need to redefine the conditions at the mass flow inlets.

- For a velocity inlet, you will specify the velocity for the individual phases. For each secondary phase, you will need to set the volume fraction (as described above). All other conditions are specified for the mixture.

- For an axis, fan, outflow, periodic, porous jump, radiator, solid, symmetry, or wall zone, all conditions are specified for the mixture. There are no conditions to be set for the individual phases. Outflow boundary conditions are not available for the cavitation model.

- For a fluid zone, mass sources are specified for the individual phases, and all other sources are specified for the mixture.

  If the fluid zone is not porous, all other conditions are specified for the mixture.

  If the fluid zone is porous, you will enable the **Porous Zone** option in the **Fluid** dialog box for the mixture. The porosity inputs (if relevant) are also specified for the mixture. The resistance coefficients and direction vectors, however, are specified separately for each phase. See User Inputs for Porous Media (p. 267) for details about these inputs. All other conditions are specified for the mixture.

See Cell Zone and Boundary Conditions (p. 237) for details about the relevant conditions for each type of boundary. Note that the pressure far-field boundary is not available with the mixture model.

Table 24.5: Phase-Specific and Mixture Conditions for the Mixture Model

Type	Primary Phase	Secondary Phase	Mixture
exhaust fan; outlet vent; pressure outlet	nothing	volume fraction specification method	all others
inlet vent; intake fan; pressure inlet	coord. system; flow direction	coord. system; flow direction; volume fraction	dir. spec. method; all others
mass flow inlet	mass flow/flux	mass flow/flux	all others
velocity inlet	velocity	velocity; volume fraction	all others
axis; fan; outflow (n/a for cavitation model); periodic;	nothing	nothing	all others

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type	Primary Phase	Secondary Phase	Mixture
porous jump; radiator; solid; symmetry; wall			
pressure far-field	not available	not available	not available
fluid	mass source; other porous inputs	mass source; other porous inputs	porous zone; porosity; all others

24.2.11.3.3. Eulerian Model

If you are using the Eulerian model, the conditions you need to specify for each type of boundary zone are listed below and summarized in

- Table 24.6: Phase-Specific and Mixture Conditions for the Eulerian Model (for Laminar Flow) (p. 1404)
- Table 24.7: Phase-Specific and Mixture Conditions for the Eulerian Model (with the Mixture Turbulence Model) (p. 1404)
- Table 24.8: Phase-Specific and Mixture Conditions for the Eulerian Model (with the Dispersed Turbulence Model) (p. 1405)
- Table 24.9: Phase-Specific and Mixture Conditions for the Eulerian Model (with the Per-Phase Turbulence Model) (p. 1405)

Note that the specification of turbulence parameters will depend on which of the three multiphase turbulence models you are using, as indicated in

- Table 24.7: Phase-Specific and Mixture Conditions for the Eulerian Model (with the Mixture Turbulence Model) (p. 1404)
- Table 24.8: Phase-Specific and Mixture Conditions for the Eulerian Model (with the Dispersed Turbulence Model) (p. 1405)
- Table 24.9: Phase-Specific and Mixture Conditions for the Eulerian Model (with the Per-Phase Turbulence Model) (p. 1405)

See Turbulence Models in the Theory Guide and Modeling Turbulence (p. 1474) for more information about multiphase turbulence models.

- For an exhaust fan, outlet vent, or pressure outlet, there are no conditions to be specified for the primary phase if you are modeling laminar flow or using the mixture turbulence model (the default multiphase turbulence model), except for backflow total temperature if heat transfer is on.

  For each secondary phase, you will need to specify the volume fraction specification method as either **Backflow Volume Fraction** or **From Neighboring Cell** (step 3 in Steps for Setting Boundary Conditions (p. 1391). If the phase is granular, you will also need to set its backflow granular temperature. If heat transfer is on, you will also need to set the backflow total temperature.

  If you are using the mixture turbulence model, you will need to specify the turbulence boundary conditions for the mixture. If you are using the dispersed turbulence model, you will need to specify them for the primary phase. If you are using the per-phase turbulence model, you will need to specify them for the primary phase and for each secondary phase.

  All other conditions are specified for the mixture.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For an inlet vent, intake fan, or pressure inlet, you will specify for the mixture which direction specification method will be used at this boundary (**Normal to Boundary** or **Direction Vector**). If you select the **Direction Vector** specification method, you will specify the coordinate system (3D only) and flow-direction components for the individual phases. If heat transfer is on, you will also need to set the total temperature for the individual phases.

  For each secondary phase, you will need to set the volume fraction (as described above). If the phase is granular, you will also need to set its granular temperature.

  If you are using the mixture turbulence model, you will need to specify the turbulence boundary conditions for the mixture. If you are using the dispersed turbulence model, you will need to specify them for the primary phase. If you are using the per-phase turbulence model, you will need to specify them for the primary phase and for each secondary phase.

  All other conditions are specified for the mixture.

- For a mass flow inlet, you will need to set the mass flow rate, mass flux, or average mass flux for each individual phase. You will also need to specify the temperature of each phase, since the energy equations are solved for each phase.

  For mass flow inlet boundary conditions, you can specify the slip velocity between phases. When you select a mass flow inlet boundary for the secondary phase, two options will be available for the **Slip Velocity Specification Method**, as shown in Figure 24.17: Mass-Flow Inlet Boundary Condition Dialog Box (p. 1403) :

  – **Velocity Ratio**

    The value for the **Velocity Ratio** is the secondary phase to primary phase velocity ratio. By default, it is 1.0, which means velocities are the same (no slip). By entering a ratio that is greater than 1.0, you are indicating a larger secondary phase velocity. Otherwise, you can enter a ratio that is less than 1.0 to indicate a smaller secondary phase velocity.

  – **Volume Fraction**

    If you specify the volume fraction at an inlet, ANSYS Fluent will calculate the phase velocities.

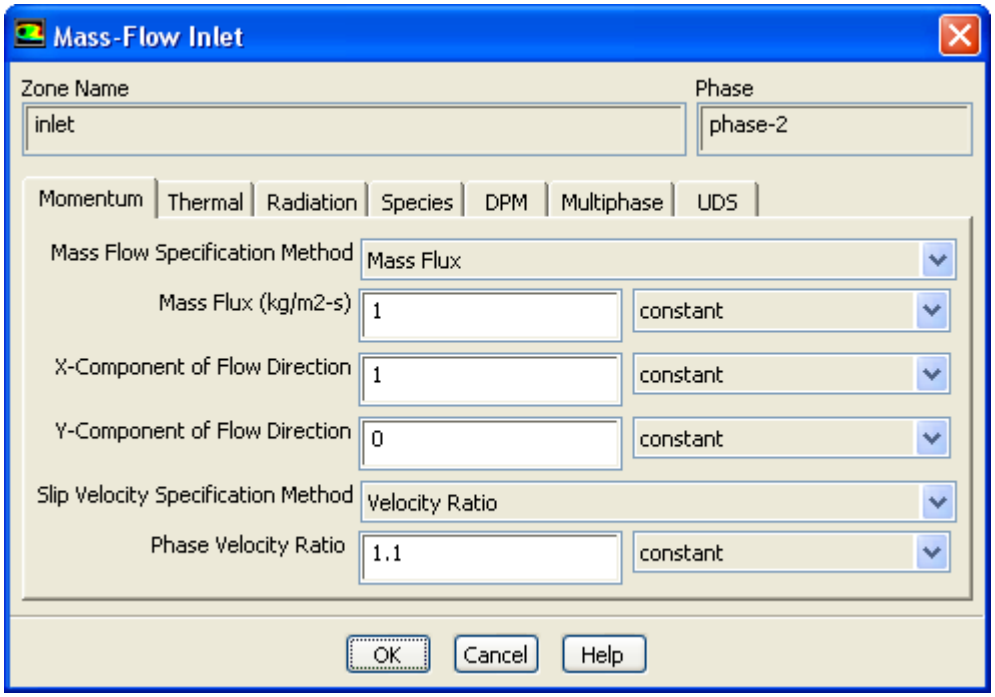
Important

– If a secondary phase has zero mass flux (that is, the Eulerian model is used to run a single phase case), neither **Velocity Ratio** nor **Volume Fraction** will affect the solution.

– The **Velocity Ratio** and **Volume Fraction** slip velocity methods cannot be mixed for different secondary phases. That is, you should specify one or the other condition for all phases.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.17: Mass-Flow Inlet Boundary Condition Dialog Box



- For a velocity inlet, you will specify the velocity for the individual phases. If heat transfer is on, you will also need to set the static temperature for the individual phases.

  For each secondary phase, you will need to set the volume fraction (as described above). If the phase is granular, you will also need to set its granular temperature.

  If you are using the mixture turbulence model, you will need to specify the turbulence boundary conditions for the mixture. If you are using the dispersed turbulence model, you will need to specify them for the primary phase. If you are using the per-phase turbulence model, you will need to specify them for the primary phase and for each secondary phase.

  All other conditions are specified for the mixture.

- For an axis, outflow, degassing, periodic, solid, or symmetry zone, all conditions are specified for the mixture. There are no conditions to be set for the individual phases.

- For a wall zone, shear conditions are specified for the individual phases. All other conditions are specified for the mixture, including thermal boundary conditions, if heat transfer is on.

- For a fluid zone, all source terms and fixed values are specified for the individual phases, unless you are using the mixture turbulence model or the dispersed turbulence model. If you are using the mixture turbulence model, source terms and fixed values for turbulence are specified instead for the mixture. If you are using the dispersed turbulence model, they are specified only for the primary phase.

  – If the fluid zone is not porous, all other conditions are specified for the mixture.

  – If the fluid zone is porous, you will enable the **Porous Zone** option in the **Fluid** dialog box for the mixture. The porosity inputs (if relevant) are also specified for the mixture. The resistance coefficients and direction vectors, however, are specified separately for each phase. See User Inputs for Porous Media (p. 267) for details about these inputs. All other conditions are specified for the mixture.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See Cell Zone and Boundary Conditions (p. 237) for details about the relevant conditions for each type of boundary. Note that the pressure far-field, fan, porous jump, radiator, and mass flow inlet boundaries are not available with the Eulerian model.

Table 24.6: Phase-Specific and Mixture Conditions for the Eulerian Model (for Laminar Flow)

Type	Primary Phase	Secondary Phase	Mixture
exhaust fan; outlet vent; pressure outlet	(tot. temperature)	volume fraction specification method; gran. temperature (tot. temperature)	all others
inlet vent; intake fan; pressure inlet	coord. system; flow direction (tot. temperature)	coord. system; flow direction; volume fraction; gran. temperature (tot. temperature)	dir. spec. method; all others
velocity inlet	velocity (tot. temperature)	velocity; volume fraction; gran. temperature (tot. temperature)	all others
mass flow inlet	mass flow rate/flux (temperature)	mass flow rate/flux; velocity ratio/volume fraction; gran. temperature (temperature)	all others
axis; outflow; degassing; periodic; solid; symmetry	nothing	nothing	all others
wall	shear condition	shear condition	all others
pressure far-field; fan; porous jump; radiator	not available	not available	not available
fluid	all source terms; all fixed values; other porous inputs	all source terms; all fixed values; other porous inputs	porous zone; porosity; all

Table 24.7: Phase-Specific and Mixture Conditions for the Eulerian Model (with the Mixture Turbulence Model)

Type	Primary Phase	Secondary Phase	Mixture
exhaust fan; outlet vent; pressure outlet	(tot. temperature)	volume fraction specification method; gran. temperature (tot. temperature)	all others
inlet vent; intake fan; pressure inlet	coord. system; flow direction (tot. temperature)	coord. system; flow direction; volume fraction; gran. temperature (tot. temperature)	dir. spec. method; all others
velocity inlet	velocity (tot. temperature)	velocity; volume fraction; gran. temperature (tot. temperature)	all others
mass flow inlet	mass flow rate/flux (temperature)	mass flow rate/flux; velocity ratio/volume fraction; gran. temperature (temperature)	all others
axis; outflow; degassing; periodic; solid; symmetry	nothing	nothing	all others
wall	shear condition	shear condition	all others

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type	Primary Phase	Secondary Phase	Mixture
pressure far-field; fan; porous jump; radiator	not available	not available	not available
fluid	other source terms; other fixed values; other porous inputs	other source terms; other fixed values; other porous inputs	source terms for turbulence; fixed values for turbulence; porous zone; porosity; all

Table 24.8: Phase-Specific and Mixture Conditions for the Eulerian Model (with the Dispersed Turbulence Model)

Type	Primary Phase	Secondary Phase	Mixture
exhaust fan; outlet vent; pressure outlet	turb. parameters (tot. temperature)	volume fraction specification method; gran. temperature (tot. temperature)	all others
inlet vent; intake fan; pressure inlet	coord. system; flow direction; turb. parameters; (tot. temperature)	coord. system; flow direction; volume fraction; gran. temperature (tot. temperature)	dir. spec. method; all others
velocity inlet	velocity; turb. parameters (tot. temperature)	velocity; volume fraction; gran. temperature (tot. temperature)	all others
mass flow inlet	mass flow rate/flux; turb. parameters (temperature)	mass flow rate/flux; velocity ratio/volume fraction; gran. temperature (temperature)	all others
axis; outflow; degassing; periodic; solid; symmetry	nothing	nothing	all others
wall	shear condition	shear condition	all others
pressure far-field; fan; porous jump; radiator	not available	not available	not available
fluid	momentum, mass, turb. sources; momentum, mass, turb. fixed values; other porous inputs	momentum and mass sources; momentum and mass fixed values; other porous inputs	porous zone; porosity; all

Table 24.9: Phase-Specific and Mixture Conditions for the Eulerian Model (with the Per-Phase Turbulence Model)

Type	Primary Phase	Secondary Phase	Mixture
exhaust fan; outlet vent; pressure outlet	turb. parameters (tot. temperature)	volume fraction specification method; turb. parameters; gran. temperature (tot. temperature)	all others
inlet vent; intake fan; pressure inlet	coord. system; flow direction; turb. parameters (tot. temperature)	coord. system; flow direction; volume fraction; turb. parameters; gran. temperature (tot. temperature)	dir. spec. method; all others

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type	Primary Phase	Secondary Phase	Mixture
velocity inlet	velocity; turb. parameters (tot. temperature)	velocity; volume fraction; turb. parameters; gran. temperature (tot. temperature)	all others
mass flow inlet	mass flow rate/flux; tub. parameters (temperature)	mass flow rate/flux; velocity ratio/volume fraction; gran. temperature (temperature)	all others
axis; outflow; degassing; periodic; solid; symmetry	nothing	nothing	all others
wall	shear condition	shear condition	all others
pressure far-field; fan; porous jump; radiator	not available	not available	not available
fluid	momentum, mass, turb. sources; momentum, mass, turb. fixed values; other porous inputs	momentum, mass, turb. sources; momentum, mass, turb. fixed values; other porous inputs	porous zone; porosity; all others

24.2.11.4. Steps for Copying Cell Zone and Boundary Conditions

The steps for copying cell zone and boundary conditions for a multiphase flow are slightly different from those described in Copying Cell Zone and Boundary Conditions (p. 242) for a single-phase flow. The modified steps are listed below:

1. In the **Cell Zone Conditions** or **Boundary Conditions** task page, click the **Copy...** button. This will open the **Copy Conditions** dialog box.

2. In the **From Cell Zone** or **From Boundary Zone** list, select the zone that has the conditions you want to copy.

3. In the **To Cell Zones** or **To Boundary Zones** list, select the zone or zones to which you want to copy the conditions.

4. In the **Phase** drop-down list, select the phase for which you want to copy the conditions (either **mixture** or one of the individual phases).

   Important

   Note that copying the boundary conditions for one phase does not automatically result in the boundary conditions for the other phases and the mixture being copied as well. You need to copy the conditions for each phase on each boundary of interest.

5. Click **Copy**. ANSYS Fluent will set *all* of the selected phase's (or mixture's) boundary conditions on the zones selected in the **To Cell Zones** or **To Boundary Zones** list to be the same as that phase's conditions on the zone selected in the **From Cell Zone** or **From Boundary Zone** list. (You cannot copy a subset of the conditions, such as only the thermal conditions.)

See Copying Cell Zone and Boundary Conditions (p. 242) for additional information about copying boundary conditions, including limitations.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.2.12. Setting Initial Volume Fractions

Once you have initialized the flow (as described in Initializing the Solution (p. 1591)), you can define the initial distribution of the phases. For a transient simulation, this distribution will serve as the initial condition at $t=0$; for a steady-state simulation, setting an initial distribution can provide added stability in the early stages of the calculation.

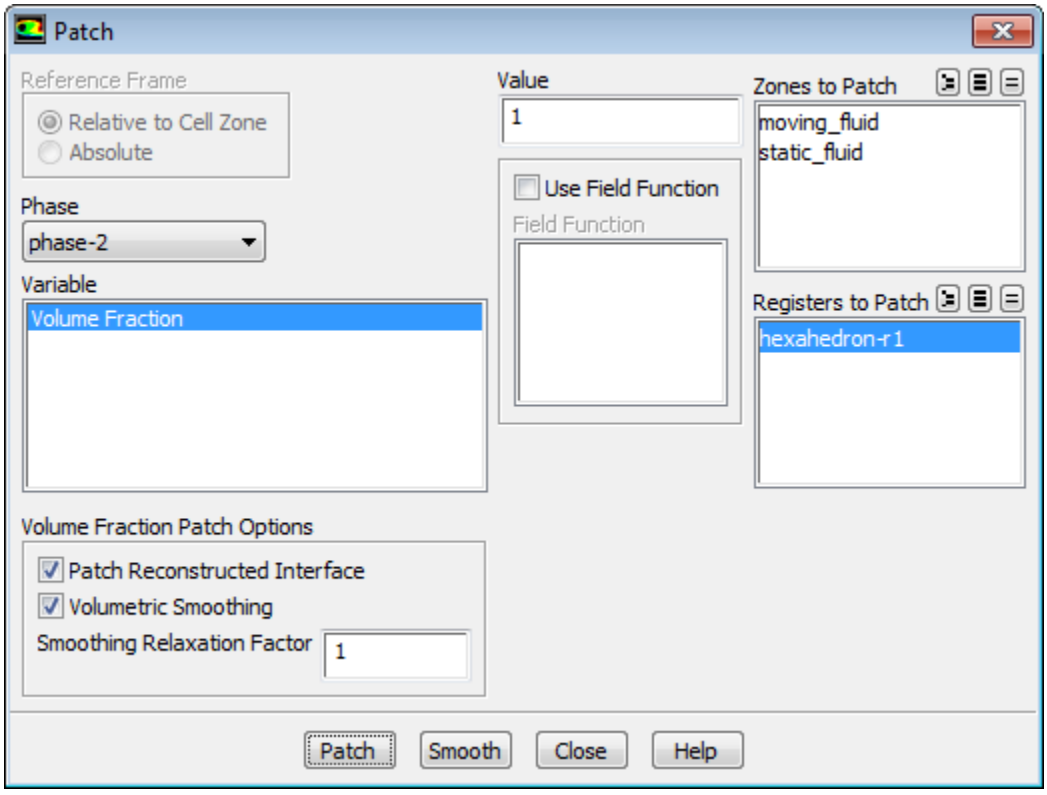
You can patch an initial volume fraction for each secondary phase using the Patch Dialog Box (p. 2522).

Solution → Initialization → Patch...

If the region in which you want to patch the volume fraction is defined as a separate cell zone, you can simply patch the value there. Otherwise, you can create a cell "register" that contains the appropriate cells and patch the value in the register. See Patching Values in Selected Cells (p. 1593) for details.

24.2.12.1. Options for Patching Volume Fraction in the VOF Model

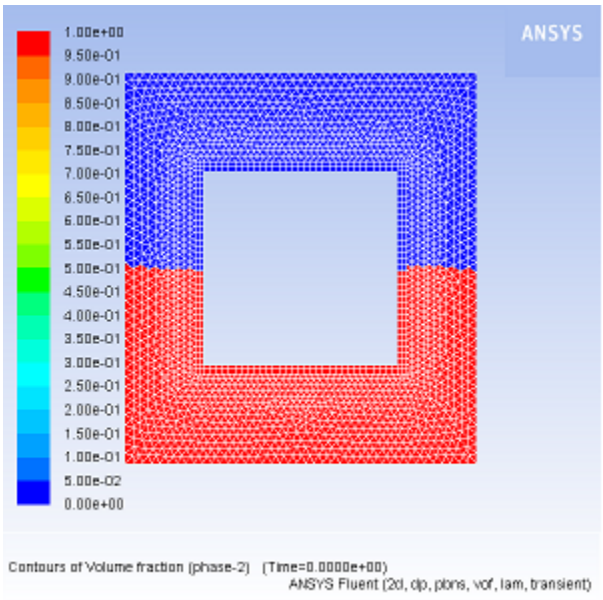
If you are using the VOF model or the Eulerian model with Multi-Fluid VOF enabled, you have additional options for how the volume fraction is patched in a user-defined register. By default the patching procedure will initialize all of the cells in the register with the specified volume fraction, without any special treatment for cells that are intersected by the fluid-fluid interface.



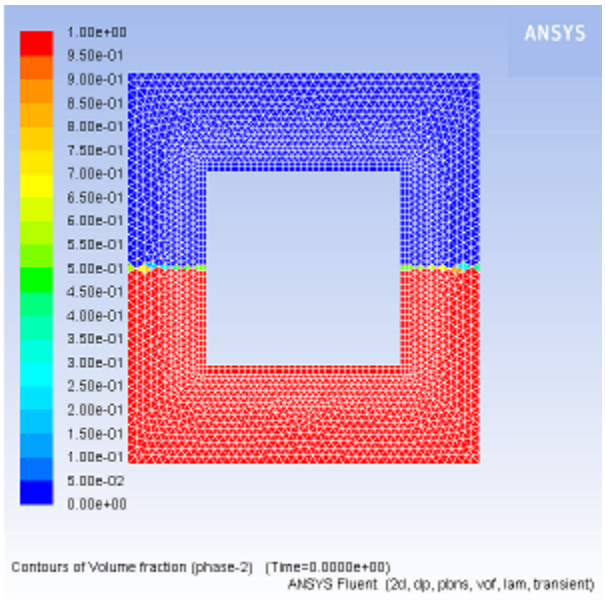
Patch Reconstructed Interface

This option uses piecewise-linear interface reconstruction to identify the fluid-fluid interface and patches the actual volume fraction field in the cells that intersect the interface.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Without Patch Reconstructed Interface



With Patch Reconstructed Interface

Important

The **Patch Reconstructed Interface** algorithm requires that the register coordinates in the **Region Adaption** dialog box be well-ordered. That is: **X Min** < **X Max**; **Y Min** < **Y Max**; **Z Min** < **Z Max**. In particular, if you use the **Select Points with Mouse** feature to specify the coordinates, be sure that this condition is met.

The **Patch Reconstructed Interface** will be applied only on the last register that was defined, even if multiple registers are selected in the **Patch** dialog box. Therefore, it is recommended that you create and patch registers one at a time.

Volumetric Smoothing

This option smooths the volume fraction field on-demand by taking the volumetric average over the cell neighbors. This can help provide better solution stability at startup. After enabling **Volumetric Smoothing**, you can click **Smooth** to apply the smoothing. Note that the smoothing is applied to the volume fraction field for all phases in the full domain. The **Smoothing Relaxation Factor** controls the degree of smoothing applied. You can specify a value between 0 (no smoothing) and 1 (maximum smoothing). For most cases, the default value is recommended. For fine meshes you may choose to try a larger value to aid solution startup. However, excessively large values can have an undesirable effect during the solution as interface sharpening discretization schemes may deteriorate the initial field.

The **Smooth** command can be used at any time after the solution has been initialized. It can therefore also be used after reading a data file for postprocessing. However, note that the smoothed field will overwrite the original volume fraction field in this case. Also note that you must select a secondary phase variable from the drop-down list in the Patch dialog box to expose the **Volumetric Smoothing** option.

24.3. Setting Up the VOF Model

For background information about the VOF model and the limitations that apply, refer to Overview of the VOF Model in the Theory Guide.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This section is organized as follows:

24.3.1. Solving Steady-State VOF Problems
24.3.2. Guidelines for Using the Multiphase Pseudo Transient Solver
24.3.3. Including Coupled Level Set with the VOF Model
24.3.4. Modeling Open Channel Flows
24.3.5. Modeling Open Channel Wave Boundary Conditions
24.3.6. Recommendations for Open Channel Initialization
24.3.7. Numerical Beach Treatment for Open Channels
24.3.8. Defining the Phases for the VOF Model
24.3.9. Setting Time-Dependent Parameters for the Explicit Volume Fraction Formulation
24.3.10. Modeling Solidification/Melting

24.3.1. Solving Steady-State VOF Problems

For steady-state VOF problems, ANSYS Fluent automatically enables the **Pseudo Transient** solution method along with the **Coupled** pressure-velocity coupling scheme for better stability and faster convergence. For more information about pseudo transient solutions, see Performing Pseudo Transient Calculations (p. 1601).

24.3.2. Guidelines for Using the Multiphase Pseudo Transient Solver

When using the **Pseudo Transient** solution method in multiphase applications, follow the guidelines provided below:

- For the fixed (**User Defined**) time step method, you must provide a suitable pseudo time-step size to the solver. You can use the **Automatic** time step method to estimate the pseudo time step size for the simulation. This can be done by setting the **Verbosity** to 1 and simulating your case for a few iterations. The pseudo-time step size will be printed in the Fluent console.
- For the **Automatic** time step method, decreasing the **Time Scale Factor** to a lower value (about 0.3) will provide better solution stability.
- Sometimes, for stiff flow problems, convergence may not be achieved when using the **Automatic** time step method. In such cases, consider switching to the fixed (**User Defined**) time step method instead.
- For steady state problems, you should never evaluate convergence based on residuals alone. To better judge convergence, you should also monitor appropriate field variables at a particular location until the value stops changing.
- The **Coupled with Volume Fractions** option is recommended for open channel flow applications.

24.3.3. Including Coupled Level Set with the VOF Model

When using the VOF formulation, you can couple the level set method with it to help overcome some limitations that exist in the interface tracking method of the VOF model and the level set method. To use the coupled level set method with VOF, perform the following:

1. Open the **Multiphase Model** dialog box.

   Setup → **Models** → **Multiphase** → **Edit...**

2. Under **Model**, enable **Volume of Fluid**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Under **Coupled Level Set + VOF**, enable **Level Set** (see Figure 24.1: Multiphase Model Dialog Box for the VOF Model (p. 1368)).

4. (optional) In some cases, spurious currents can arise from the application of the surface tension force in the momentum equation. In these cases you can use the `/define/models/multiphase/coupled-level-set` text command to use a revised formulation of the surface tension force that weights the force towards the heavier phase in the interface cells. You can choose from `none` (the default), `density-correction`, and `heaviside-correction`.

After the **Level Set** option is enabled, proceed as you normally would when setting up the VOF model (described in Setting Up the VOF Model (p. 1408)). For theoretical information, refer to Coupled Level-Set and VOF Model.

Note

When using the **Level Set** option, the recommended scheme is the geo-reconstruct scheme (see The Geometric Reconstruction Scheme).

Important

- The level set method is only suitable for two-phase flow regime, where two fluids are not interpenetrating.
- The level set model can only be used when the VOF model is turned on. No mass transfer is allowed.
- The level set method is not compatible with the dynamic mesh model.

Normally, zero flux of the level set function is set as the default for the boundary conditions. Due to the geometrical re-initialization procedure at each time step, the boundary conditions shall not have any significant effect on the results. For more information, see Re-initialization of the Level-set Function via the Geometrical Method.

24.3.4. Modeling Open Channel Flows

Using the VOF formulation, open channel flows can be modeled in ANSYS Fluent. To start using the open channel flow boundary condition, perform the following:

1. Enable **Gravity** and set the gravitational acceleration fields.

   Setup → **General** **Gravity** → **On**

2. Enable the volume of fluid model.

   a. Open the **Multiphase Model** dialog box.

      Setup → **Models** → **Multiphase** → **Edit...**

   b. Under **Model**, enable **Volume of Fluid**.

   c. Under **Formulation**, select either **Implicit** or **Explicit**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Under **VOF Sub-Models**, select **Open Channel Flow**.

> **Note**
>
> The default VOF formulation is set to **Implicit** after enabling the **Open Channel Flow** option. This is done to allow the use of larger time step sizes for such applications.

In order to set specific parameters for a particular boundary for open channel flows, enable the **Open Channel** option in the **Multiphase** tab of the corresponding boundary condition dialog box. Table 24.10: Open Channel Boundary Parameters for the VOF Model (p. 1411) summarizes the types of boundaries available to the open channel flow boundary condition, and the additional parameters needed to model open channel flow. For more information on setting boundary condition parameters, see Cell Zone and Boundary Conditions (p. 237).

Table 24.10: Open Channel Boundary Parameters for the VOF Model

Boundary Type	Parameter
pressure inlet	Inlet Group ID; Secondary Phase for Inlet; Flow Specification Method; Free Surface Level, Bottom Level; Velocity Magnitude
pressure outlet	Outlet Group ID; Pressure Specification Method; Free Surface Level; Bottom Level
mass flow inlet	Inlet Group ID; Secondary Phase for Inlet; Free Surface Level; Bottom Level; Mass Flow Rates for the Phases
velocity inlet	Secondary Phase for Inlet; Free Surface Level, Averaged Velocity Inputs/Segregated Velocity Inputs (Available with Open Channel Wave Boundary Condition)
outflow	Flow Rate Weighting

24.3.4.1. Defining Inlet Groups

Open channel systems involve the flowing fluid (the secondary phase) and the fluid above it (the primary phase).

If both phases enter through the separate inlets (for example, `inlet-phase2` and `inlet-phase1`), these two inlets form an inlet group. This inlet group is recognized by the parameter **Inlet Group ID**, which will be same for both the inlets that make up the inlet group. On the other hand, if both the phases enter through the same inlet (for example, `inlet-combined`), then the inlet itself represents the inlet group.

> **Important**
>
> In three-phase flows, only one secondary phase is allowed to pass through one inlet group.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.3.4.2. Defining Outlet Groups

Outlet-groups can be defined in the same manner as the inlet groups.

Important

In three-phase flows, the outlet should represent the outlet group, that is, separate outlets for each phase are not recommended in three-phase flows.

24.3.4.3. Setting the Inlet Group

For pressure inlets and mass flow inlets, the **Inlet Group ID** is used to identify the different inlets that are part of the same inlet group. For instance, when both phases enter through the same inlet (single face zone), then those phases are part of one inlet group and you would set the **Inlet Group ID** to 1 for that inlet (or inlet group).

In the case where the same inlet group has separate inlets (different face zones) for each phase, then the **Inlet Group ID** will be the same for each inlet of that group.

When specifying the inlet group, use the following guidelines:

- Since the **Inlet Group ID** is used to identify the inlets of the same inlet group, general information such as **Free Surface Level**, **Bottom Level**, or the mass flow rate for each phase should be the same for each inlet of the same inlet group.
- You should specify a different **Inlet Group ID** for each distinct inlet group.

  For example, consider the case of two inlet groups for a particular problem. The first inlet group consists of water and air entering through the same inlet (a single face zone). In this case, you would specify an inlet group ID of 1 for that inlet (or inlet group). The second inlet group consists of oil and air entering through the same inlet group, but each uses a different inlet (`oil-inlet` and `air-inlet`) for each phase. In this case, you would specify the same **Inlet Group ID** of 2 for both of the inlets that belong to the inlet group.

24.3.4.4. Setting the Outlet Group

For pressure outlet boundaries, the **Outlet Group ID** is used to identify the different outlets that are part of the same outlet group. For instance, when both phases enter through the same outlet (single face zone), then those phases are part of one outlet group and you would set the **Outlet Group ID** to 1 for that outlet (or outlet group).

In the case where the same outlet group has separate outlets (different face zones) for each phase, then the **Outlet Group ID** will be the same for each outlet of that group.

When specifying the outlet group, use the following guidelines:

- Since the **Outlet Group ID** is used to identify the outlets of the same outlet group, general information such as **Free Surface Level** or **Bottom Level** should be the same for each outlet of the same outlet group.
- You should specify a different **Outlet Group ID** for each distinct outlet group.

  For example, consider the case of two outlet groups for a particular problem. The first outlet group consists of water and air exiting from the same outlet (a single face zone). In this case, you would

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

specify an outlet number of 1 for that outlet (or outlet group). The second outlet group consists of oil and air exiting through the same outlet group, but each uses a different outlet (`oil-outlet` and `air-outlet`) for each phase. In this case, you would specify the same **Outlet Group ID** of 2 for both of the outlets that belong to the outlet group.

Important

For three-phase flows, when all the phases are leaving through the same outlet, the outlet should consist only of a single face zone.

24.3.4.5. Determining the Free Surface Level

For the appropriate boundary, you need to specify the **Free Surface Level** value. This parameter is available for all relevant boundaries, including pressure outlet, mass flow inlet, and pressure inlet. The **Free Surface Level**, is represented by y_{local} in Equation 17.41 in the Theory Guide.

$$y_{local} = -\left(\vec{a} \cdot \hat{g}\right) \tag{24.2}$$

where $\vec{a}$ is the position vector of any point on the free surface, and $\hat{g}$ is the unit vector in the direction of the force of gravity. Here a horizontal free surface that is normal to the direction of gravity is assumed.

We can simply calculate the free surface level in two steps:

1. Determine the absolute value of height from the free surface to the origin in the direction of gravity.

2. Apply the correct sign based on whether the free surface level is above or below the origin.

If the liquid's free surface level lies above the origin, then the **Free Surface Level** is positive (see Figure 24.18: Determining the Free Surface Level and the Bottom Level (p. 1414)). Likewise, if the liquid's free surface level lies below the origin, then the **Free Surface Level** is negative.

You can also specify a transient profiles for a **Free Surface Level** for the relevant open channel boundaries as shown in the example below:

```
/*********************************************************************
Example UDF that demonstrates transient profile for free surface level
*********************************************************************/

#include "udf.h"

#define H 1.5   /* Original Free surface level */
#define T0 0.2  /* Time */

DEFINE_TRANSIENT_PROFILE(fs_level, current_time)
{
    real level;
    if (current_time <= T0)
      level = H - current_time;
    else
      level = H - T0;

    return level;
}
```

24.3.4.6. Determining the Bottom Level

For the appropriate boundary, you need to specify the **Bottom Level** value. This parameter is available for all relevant boundaries, including pressure outlet, mass flow inlet, and pressure inlet. The **Bottom Level**, is represented by a relation similar to Equation 17.41 in the Theory Guide.

$$y_{bottom} = -\left(\vec{b} \cdot \hat{g}\right) \tag{24.3}$$

where $\vec{b}$ is the position vector of any point on the bottom of the channel, and $\hat{g}$ is the unit vector of gravity. Here we assume a horizontal free surface that is normal to the direction of gravity.

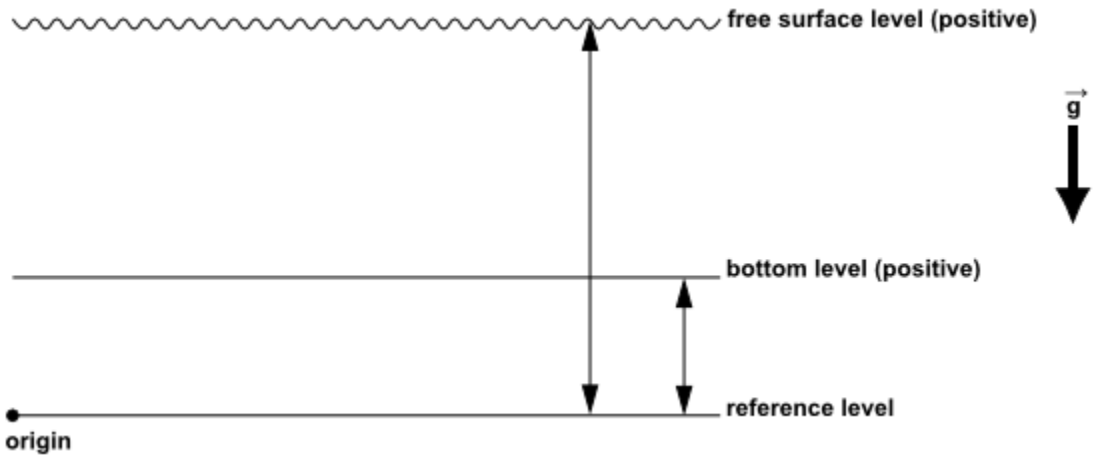
We can simply calculate the bottom level in two steps:

1. Determine the absolute value of depth from the bottom level to the origin in the direction of gravity.

2. Apply the correct sign based on whether the bottom level is above or below the origin.

If the channel's bottom lies above the origin, then the **Bottom Level** is positive (see Figure 24.18: Determining the Free Surface Level and the Bottom Level (p. 1414)). Likewise, if the channel's bottom lies below the origin, then the **Bottom Level** is negative.

Figure 24.18: Determining the Free Surface Level and the Bottom Level



You can also specify a transient profiles for a **Bottom Level** for the relevant open channel boundaries.

24.3.4.7. Specifying the Total Height

The total height, along with the velocity, is used as an option for describing the flow. The total height is given as

$$y_{tot} = y_{local} + \frac{V^2}{2g} \tag{24.4}$$

where V is the velocity magnitude and g is the gravity magnitude.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.3.4.8. Determining the Velocity Magnitude

For pressure inlet boundaries, input for **Velocity Magnitude** is required to calculate the dynamic pressure being used in the total pressure calculation.

Note

The provided **Velocity Magnitude** is not applied to the boundary.

24.3.4.9. Determining the Secondary Phase for the Inlet

For pressure inlets and mass flow inlets, the **Secondary Phase for Inlet** field allows you to choose the desired secondary phase in the case of three-phase flows.

Important

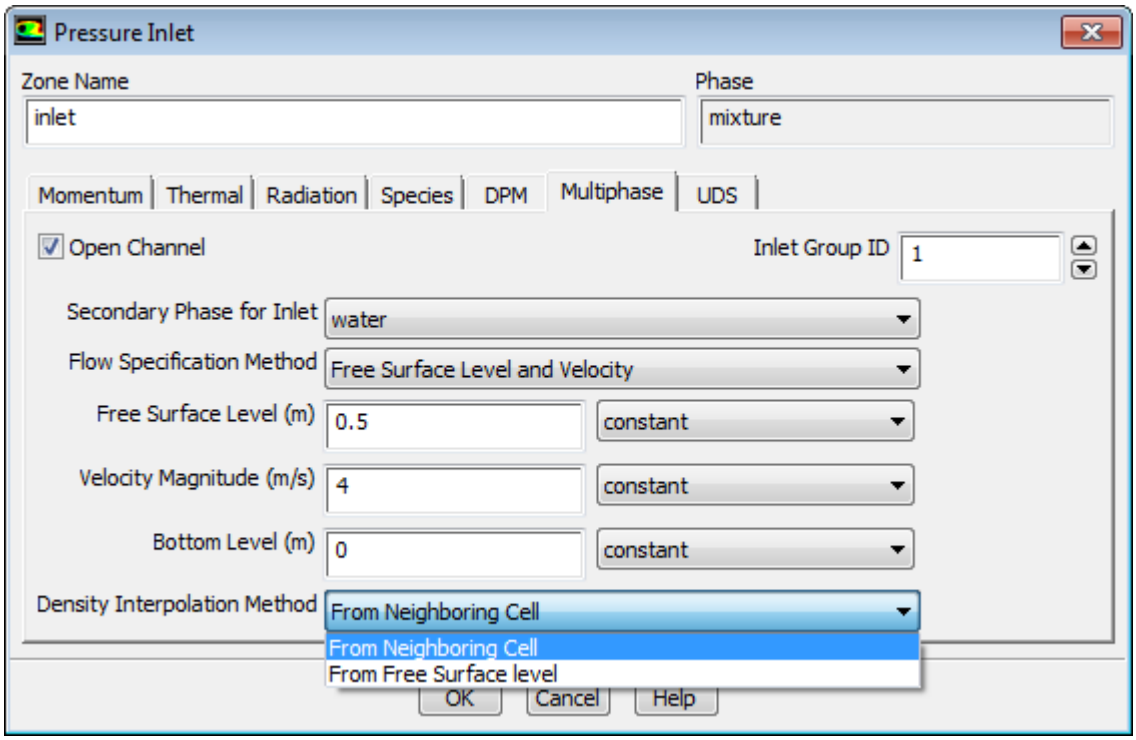
Note that only one secondary phase is allowed to pass through one inlet group.

Consider a problem involving a three-phase flow consisting of air as the primary phase, and oil and water as the secondary phases. Consider also that there are two inlet groups:

- water and air
- oil and air

For the former inlet group, you would choose water as the secondary phase. For the latter inlet group, you would choose oil as the secondary phase (as shown in Figure 24.19: Pressure Inlet for Open Channel Flow (p. 1415)).

Figure 24.19: Pressure Inlet for Open Channel Flow



24.3.4.10. Choosing the Pressure Specification Method

For a pressure outlet boundary, the outlet pressure can be specified in one of three ways:

- by prescribing the free surface level, such as a hydrostatic pressure profile (available for two-phase flow only)
- by specifying the constant pressure
- by specifying the neighboring cell pressure

Note

You can also specify a hydrostatic pressure profile at the pressure outlet for axisymmetric open channel flow. However, there are certain limitations as noted in Limitations (p. 1417).

Important

This option is not available in the case of three-phase flows since the pressure on the boundary is taken from the neighboring cell.

24.3.4.11. Choosing the Density Interpolation Method

For problems involving sub-critical flow, the following options exist for the **Density Interpolation Method**:

- **From Neighboring Cell** is the default option, where the mixture density used in the hydrostatic profile is interpolated using the volume fraction calculated from the neighboring cell.

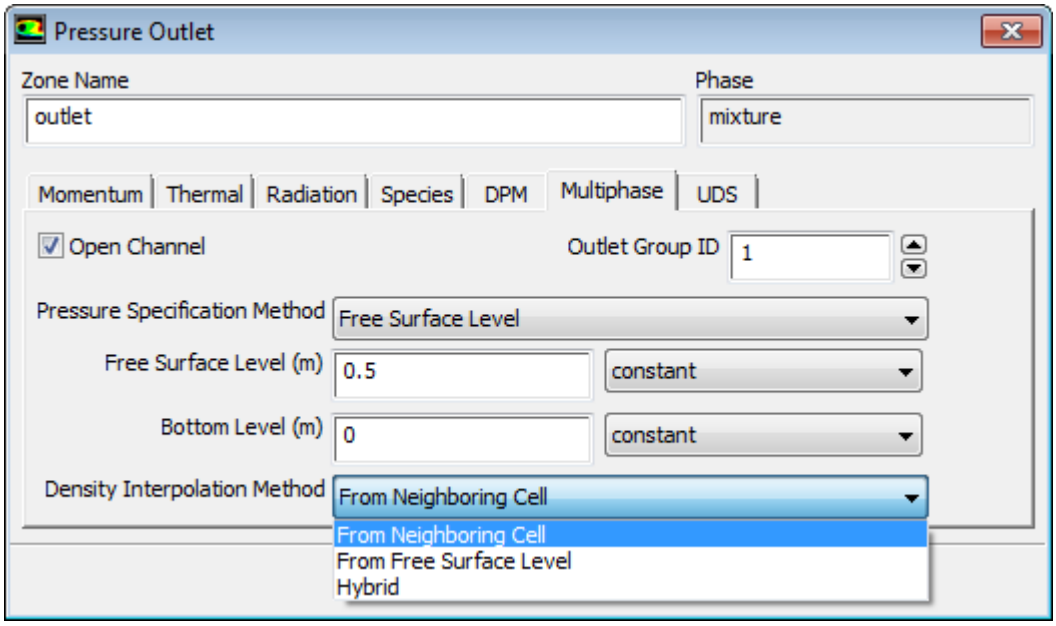
  Note

  In some cases, this method may cause free surface oscillation near the upstream boundaries. These oscillations are particularly visible in cases where boundaries are close to the object; or if the boundary has unstructured mesh near the free surface level. For such cases, it is recommended that you use the `From Free Surface Level` option under **Density Interpolation Method**.

- **From Free Surface Level** where the mixture density used in the hydrostatic profile is interpolated from the volume fraction calculated from the free surface level.
- **Hybrid** where the mixture density used in the hydrostatic profile is calculated either from the **From Neighboring Cell** or **From Free Surface Level** options, depending on the direction of flux. This method is only available at a pressure outlet.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.20: Density Interpolation Method for Open Channel Flow



24.3.4.12. Open Channel Flow Compatibility with Velocity Inlet

For additional information, see the following sections:
24.3.4.12.1. Velocity Inlet, Open Channel Flow, Steady-State
24.3.4.12.2. Velocity Inlet, Open Channel Flow, Transient

Open channel flow compatibility with velocity inlet is provided in the framework of the open channel wave boundary condition, therefore, the **Open Channel Wave BC** option in the Figure 24.1: Multiphase Model Dialog Box for the VOF Model (p. 1368) must be enabled first.

Velocity inlet compatibility allows you to use either averaged or segregated velocity inputs to moving obstacles, water and air, which is not possible using either the pressure inlet or mass flow inlet boundary conditions.

Select the **Open Channel Wave BC** option in the **Velocity Inlet** dialog box. For detailed information about the velocity inputs, see Inputs at Velocity Inlet Boundaries (p. 317).

24.3.4.12.1. Velocity Inlet, Open Channel Flow, Steady-State

you must specify (in the **Multiphase** tab of the **Velocity Inlet** dialog box):

- Secondary phase for the inlet
- Free surface level

24.3.4.12.2. Velocity Inlet, Open Channel Flow, Transient

You must set the **Wave BC Option** to **None** (in the **Multiphase** tab of the **Velocity Inlet** dialog box).

24.3.4.13. Limitations

The following list summarizes some issues and limitations associated with the open channel boundary condition.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The conservation of the Bernoulli integral does not provide the conservation of mass flow rate for the pressure boundary. In the case of a coarser mesh, there can be a significant difference in mass flow rate from the actual mass flow rate. For finer meshes, the mass flow rate comes closer to the actual value. So, for problems having constant mass flow rate, the mass flow rate boundary condition is a better option. The pressure boundary should be selected when steady and non-oscillating drag is the main objective.
- Specifying the top boundary as the pressure outlet can sometimes lead to a divergent solution. This may be due to the corner singularity at the pressure boundary in the air region or due to the inability to specify local flow direction correctly if the air enters through the top locally.
- Only the heavier phase should be selected as the secondary phase.
- In the case of three-phase flows, only one secondary phase is allowed to enter through one inlet group (that is, the mixed inflow of different secondary phases is not allowed).
- For axisymmetric flow, pressure and mass flow inlets do not support open channel boundary conditions.
- Open channel wave boundary conditions for velocity inlets do not support cases with axisymmetric flow.
- Axisymmetric open channel flow is available for two phase flow and the pressure outlet boundary.
- For axisymmetric open channel flow, the direction of gravity must be aligned with the direction of the axis.

24.3.4.14. Recommendations for Setting Up an Open Channel Flow Problem

The following list represents a list of recommendations for solving problems using the open channel flow boundary condition:

- In the cases where the inlet group has a different inlet for each phase of fluid, then the parameter values (such as **Free Surface Level**, **Bottom Level**, and **Mass Flow Rate**) for each inlet should correspond to all other inlets that belong to the inlet group.
- The solution begins with an estimated pressure profile at the outlet boundary.

  In general, you can start the solution by assuming that the level of liquid at the outlet corresponds to the level of liquid at the inlet. The convergence and solution time is very dependent on the initial conditions. When the flow is completely subcritical (upstream and downstream), in marine applications for instance, the above approach is recommended.

  If the final conditions of the flow can be predicted by other means, the solution time can be significantly reduced by using the proper boundary condition.
- In the case of super-critical flow at the outlet, you can use the **From Neighboring Cell** option at the outlet, as it may provide better convergence.
- The initialization procedure is very critical in the open channel analysis. Refer to Recommendations for Open Channel Initialization (p. 1429).
- For the initial stability of the solution, a smaller time step is recommended. You can increase the time step once the solution becomes more stable.
- For flows that do not make a transition from sub-critical to super-critical, or vice-versa, you can speed-up the solution calculation by specifying the frequency of Froude number updates during run time by setting the following text command: `solve` → `set` → `open-channel-controls`.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When prompted, set up the following parameter for steady-state flows:

```
/solve/set> open-channel-controls
Iteration interval for Froude number update [10]
```

When prompted, set up the following parameter for transient flows:

```
/solve/set> open-channel-controls
Time step interval for Froude number update [1]
```

- For pure open channel flow applications, the inlet and outlet boundary conditions are controlled by the Froude number. In certain cases, it is possible to impose a hydrostatic profile at the outlet without any Froude number dependency. You can use the following text command: `solve` → `set` → `open-channel-controls`.

  When prompted, set up the following parameter as shown below:

  ```
  /solve/set> open-channel-controls
  Use Froude number independent boundary condition at outlet? [no] yes
  ```

- Relevant open channel inputs, along with the Froude number, can be reported using the following text command: `define` → `boundary-conditions` → `openchannel-threads`.

- In the case of reverse flow, the pressure outlet boundary behaves as a pressure inlet, and the boundary-specific static pressure is taken as the **Total Pressure**. In this case, the static pressure is calculated from the total pressure. You can use the following text command to fix the boundary-specified static pressure, which helps to suppress reflections from the pressure boundary for certain cases: `solve` → `set` → `open-channel-controls`

  When prompted, set up the following parameter as shown below:

  ```
  /solve/set> open-channel-controls
  Use boundary specified static pressure for backflow? [no] yes
  ```

24.3.5. Modeling Open Channel Wave Boundary Conditions

When modeling open channel wave boundary conditions, many of the variables that are used in open channel flow, also exist for open channel wave boundary conditions. You may have to refer to Modeling Open Channel Flows (p. 1410) for information about some of the settings.

To use the open channel wave boundary condition, perform the following:

1. Enable **Gravity** and set the gravitational acceleration fields.

   Setup → **General** **Gravity** → **On**

2. Enable the **Volume of Fluid** model in the **Multiphase Model** dialog box.

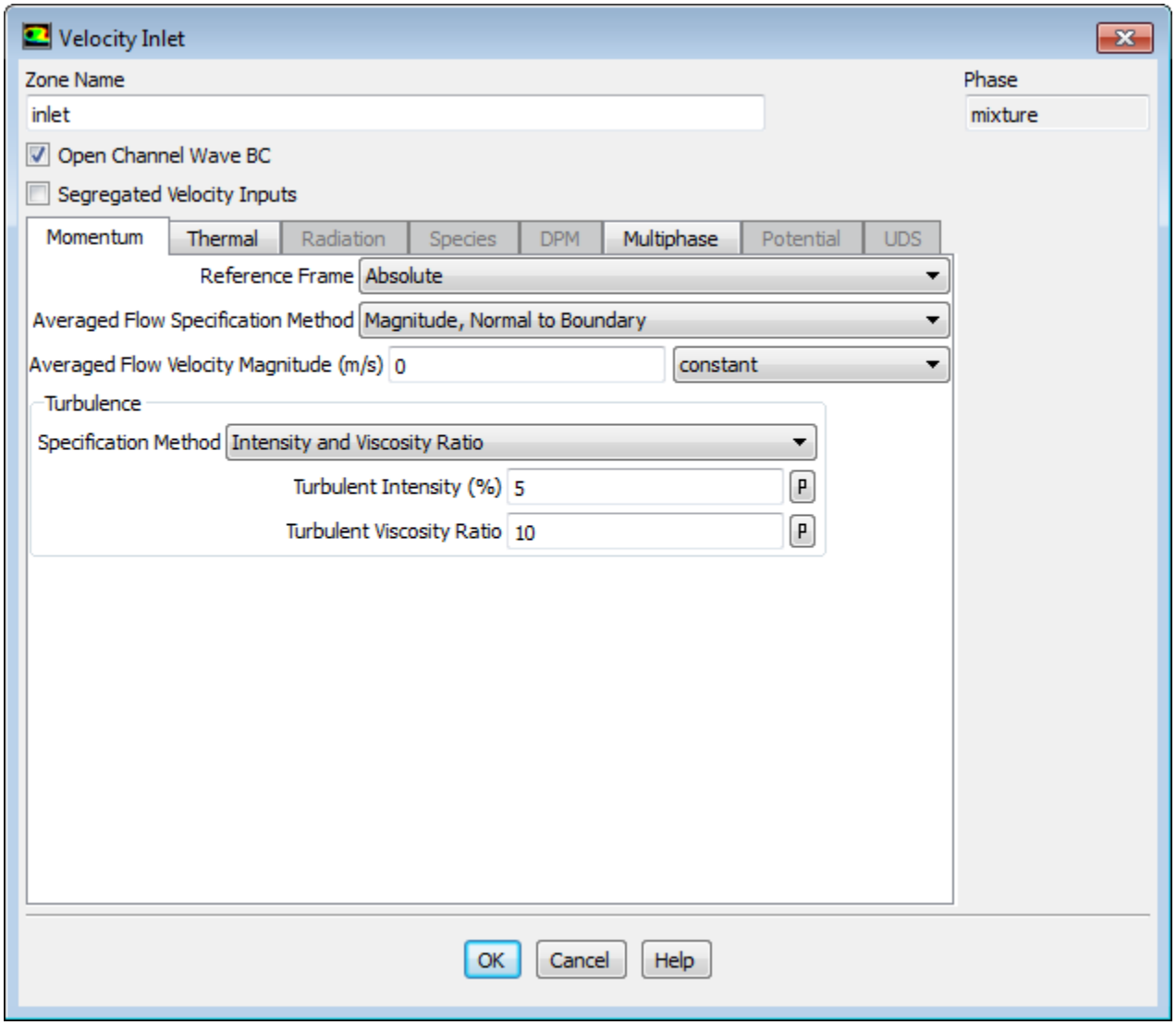
   Setup → **Models** → **Multiphase** → **Edit...**

3. Under **Formulation**, select either **Implicit** or **Explicit**.

4. Under **VOF Sub-Models**, select **Open Channel Wave BC**.

In order to set specific parameters for a particular boundary for open channel wave boundaries, enable the **Open Channel Wave BC** option in the **Velocity Inlet** boundary condition dialog box (Figure 24.21: The Velocity Inlet for Open Channel Wave BC (p. 1420)).

Figure 24.21: The Velocity Inlet for Open Channel Wave BC



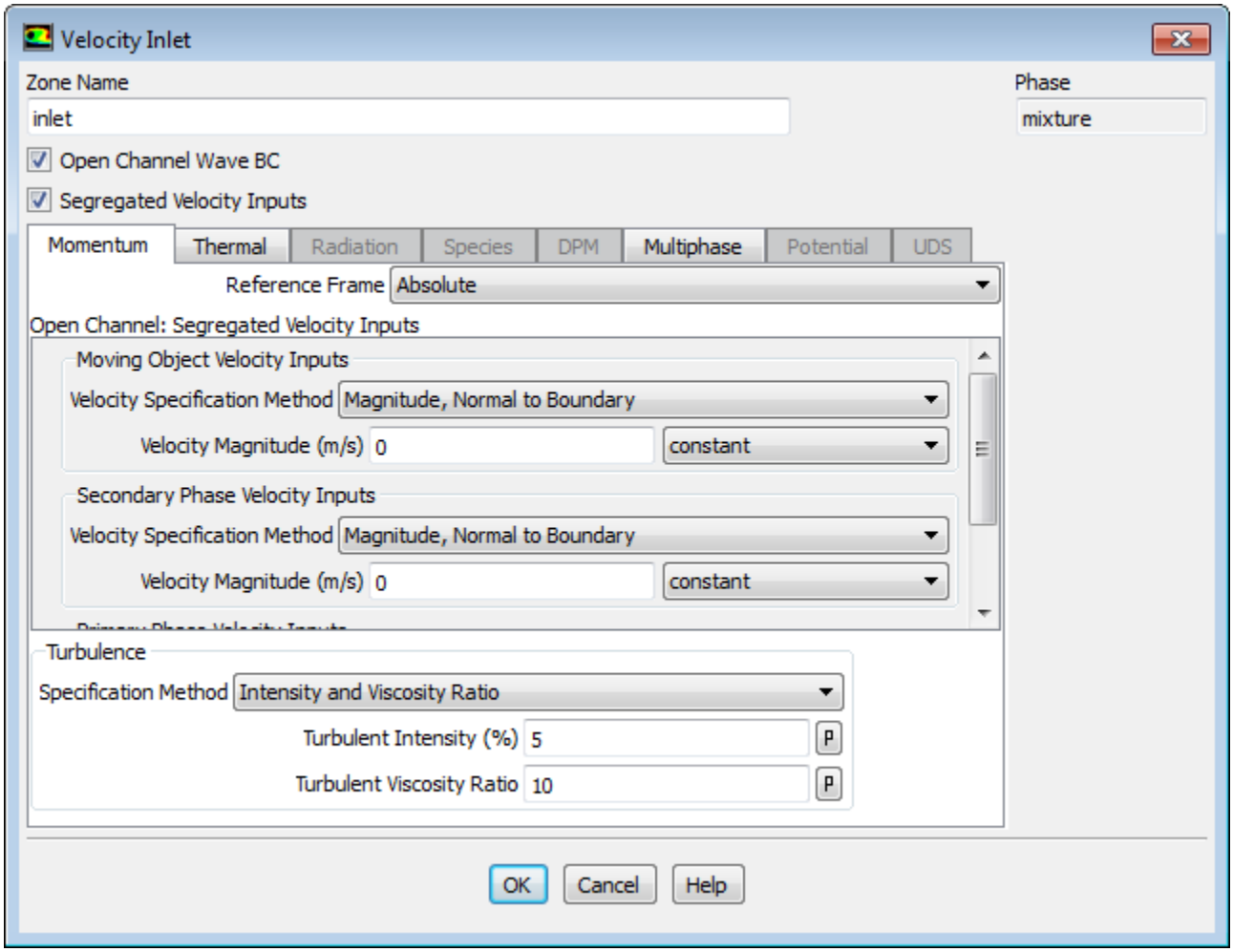
The default method is the **Averaged Flow Specification Method**, where inputs for the moving obstacle and water current are combined. There is no specific treatment for air velocity, which is assumed to be same as the averaged velocity.

Segregated Velocity Inputs

The **Segregated Velocity Inputs** option allows you to prescribe velocity inputs individually for the moving obstacle, Secondary phase (typically water), and Primary phase (typically air).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.22: Segregated Velocity Inputs for Open Channel Wave BC



You can set the moving object and secondary phase velocity specification methods as:

- Magnitude and direction vector
- Magnitude and normal to boundary

You can set the primary phase velocity specification methods as:

- Magnitude and direction vector
- Magnitude and normal to boundary
- Power law and direction vector
- Power law and normal to boundary

Power Law

$$\frac{V}{V_{ref}} = \left(\frac{z}{z_{ref}}\right)^{\alpha} \tag{24.5}$$

Where z_{ref} is the vertical distance from the instantaneous free surface level at which the reference velocity magnitude, V_{ref} is specified. z is the vertical distance from the free surface level at which,

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

velocity V must be estimated. α is the power law coefficient, which is typically 0.16 for an open sea.

Setting Buffer Layer Height

The buffer layer is assumed to be the layer above the free surface, in which the effect of air is not felt. This option is provided by assuming that the air in the vicinity of free surface blows with the speed of the water and the waves, so that direct effect of air is not felt in the wave propagation.

There are two options for providing the buffer height:

1. **Automatic:** the buffer layer is taken as the specified fraction of the maximum wave height.
2. **User-defined:** you provide the input for the buffer layer height.

The buffer layer can also be defined using the following text command:

```
/solve/set> open-channel-wave-options

/solve/set/open-channel-wave-options>
set-buffer-layer-ht     set-verbosity           stokes-wave-variants

/solve/set/open-channel-wave-options> set-buffer-layer-ht
buffer layer specification method (0 - Automatic, 1 - User-defined)  [0]
fraction of buffer layer height to maximum wave height [0.02]

/solve/set/open-channel-wave-options> set-buffer-layer-ht
buffer layer specification method (0 - Automatic, 1 - User-defined)  [0] 1
enter the value of buffer layer height [0]
```

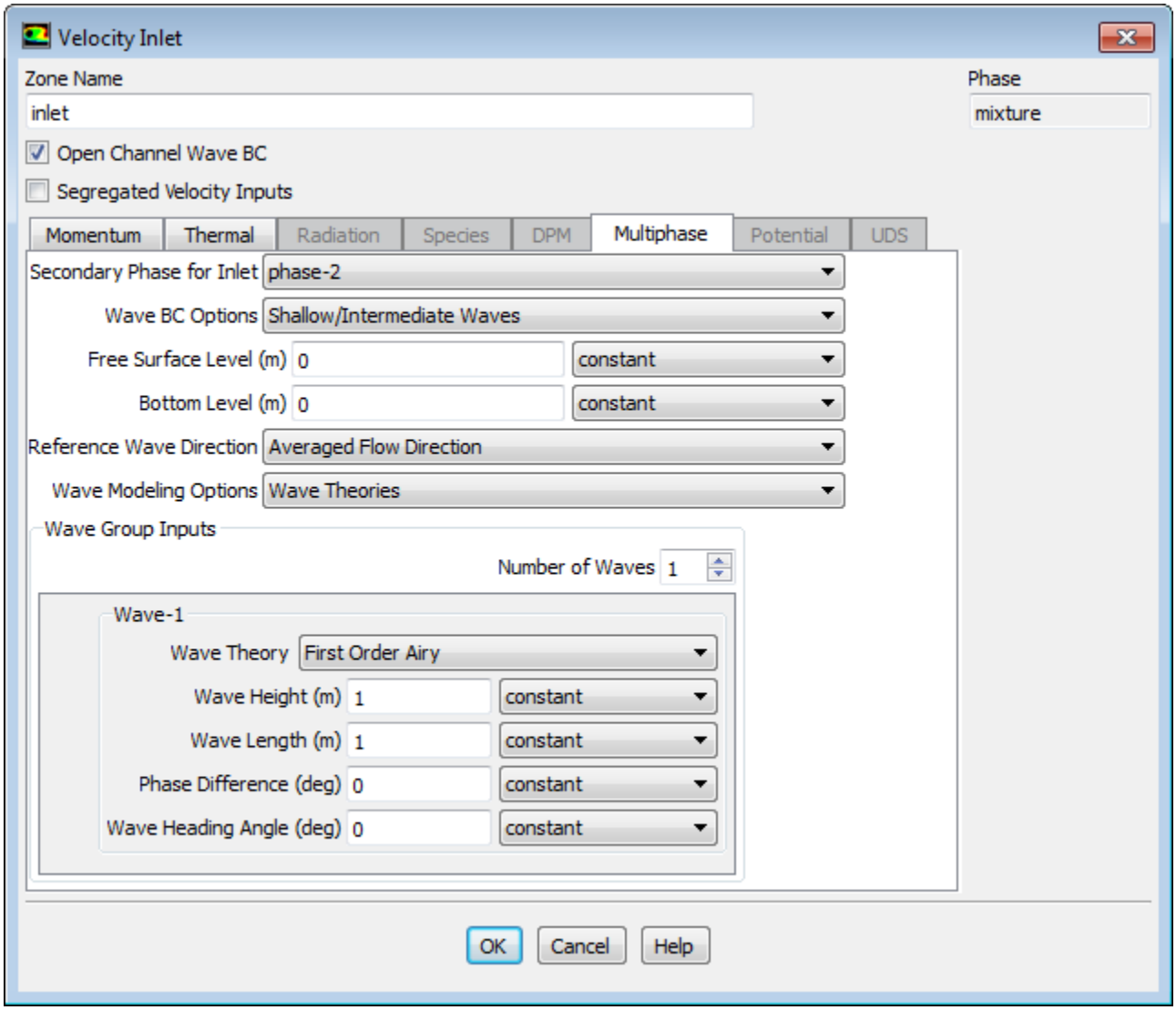
In the **Momentum** tab of the **Velocity Inlet** dialog box, you can enter the averaged flow velocity, which includes components from the flow current and the moving object. You can specify the **Averaged Flow Specification Method** as:

- **Magnitude and Direction**
- **Magnitude and Normal to Boundary**
- **Components**

In the **Multiphase** tab (Figure 24.23: The Velocity Inlet for Open Channel Wave BC (p. 1423)), you will specify the following:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.23: The Velocity Inlet for Open Channel Wave BC



- **Secondary Phase for Inlet** specifies the phase to which the wave parameters are applied. In case of a three-phase flow, select the corresponding secondary phase from this list.
- **Wave BC Options** of which you have a choice of **Shallow/Intermediate Waves**, **Short Gravity Waves**, **Shallow Waves**, and **None** (for calm sea or pure open channel flow problems without waves). Information about these waves is available in Open Channel Wave Boundary Conditions in the Theory Guide. Note that the short gravity waves expression is derived under the assumption of infinite liquid height.
- **Free Surface Level** is the same definition as for open channel flow, see Modeling Open Channel Flows (p. 1410).
- **Bottom Level** is the same definition as for open channel flow, see Modeling Open Channel Flows (p. 1410), and is valid only for shallow or intermediate depth waves. The bottom level is used for calculating the liquid height.
- **Reference Wave Direction** is the direction of wave propagation with zero wave heading angle. You can specify the wave propagation direction as:
  - **Averaged Flow Direction**: In this case, the reference direction is the same as the averaged flow direction.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **Direction Vector**
  - **Normal to Boundary**

- **Wave Modeling Options** determines how the waves are modeled in the shallow/intermediate and short gravity wave regimes. You can choose to model the waves using either **Wave Theories** or **Wave Spectrum**. **Wave Theories** is usually selected for simulating regular waves. To simulate random waves, select **Wave Spectrum**.
- Depending on your choice of **Wave BC Options** and **Wave Modeling Options**, you will need to specify the wave parameters as described in the following sections.

Shallow Wave Inputs (p. 1424)
Wave Group Inputs (p. 1425)
Wave Spectrum Inputs (p. 1425)

Shallow Wave Inputs

If you have selected **Shallow Waves** for **Wave BC Options** you can specify the following settings for each wave:

Number of Waves
is the option to set the number of interacting waves (default = 1)

Ideally, shallow waves do not support the principle of superposition. The **Number of Waves** option enables interaction of two different waves originating at unique (non-interfering) locations within the domain. Collision of two counter-propagating solitary waves inside a domain is an example of a shallow wave interaction.

Wave Theory
of which you have a choice of **Fifth Order Solitary** (the default) and **Fifth Order Cnoidal**. Information about the types of wave theory is available in Open Channel Wave Boundary Conditions in the Theory Guide.

Wave Height
is the height difference between a wave crest to the neighboring trough.

Since a solitary wave does not have troughs, the wave height is the distance between a wave crest to mean free surface level.

Wave Length
is the distance between two consecutive zero crossings.

Since solitary waves are derived based on the assumption of infinite wave length, specified wave length is only used to estimate the elliptic function parameter for suitability of the wave theory, and is not used in calculating wave parameters.

Inlet Offset Distance
is the translational distance from the reference point origin in the reference wave propagation direction. This option is used to generate a wave from a location other than the reference frame origin. For a solitary wave, the hump is always generated at the location where:

$$x - x_0 = 0 \tag{24.6}$$

where x and x_0 are spatial coordinates in the reference wave propagation direction.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Wave Heading Angle
is the angle between the direction of the wavefront and the reference wave propagation direction, in the plane of the flow surface. In 2D, there are only two possibilities, zero degree when the wave is in the reference propagation direction, and 180 degree when the wave Is in the direction opposite to the reference propagation direction.

Wave Group Inputs

If you have selected **Wave Theories** for **Wave Modeling Options** you can specify the following settings for each wave:

Number of Waves
is the option to set the number of superposed waves (default = 1)

Wave Theory
of which you have a choice of **First Order Airy** (the default), **Second Order Stokes**, **Third Order Stokes**, **Fourth Order Stokes**, and **Fifth Order Stokes**. Information about the types of wave theory is available in Open Channel Wave Boundary Conditions in the Theory Guide.

Wave Height
is the height difference between a wave crest to the neighboring trough.

Wave Length
is the distance between two consecutive crests, troughs or zero crossings.

Phase Difference
is the phase angle by which one periodic disturbance or wavefront lags behind or precedes another in time or space.

Wave Heading Angle
is the angle between the direction of the wavefront and the reference wave propagation direction, in the plane of the flow surface. In 2D, there are only two possibilities, zero degree when the wave is in the reference propagation direction, and 180 degree when the wave Is in the direction opposite to the reference propagation direction.

Wave Spectrum Inputs

If you have selected **Wave Spectrum** for **Wave Modeling Options** you can specify the following settings for each wave:

Frequency Spectrum Method
specifies the spectrum to use. You can select **Pierson-Moskowitz** (appropriate for fully-developed seas), **Jonswap** (appropriate for fetch-limited seas), or **TMA** (appropriate for fetch-limited, finite-depth seas).

Peak Shape Parameter
controls the shape and amplitude of the frequency peak in the **Jonswap** and **TMA** formulations. This corresponds to γ in Equation 17.100 in the *Fluent Theory Guide*.

Significant Wave Height
is the mean wave height of the largest 1/3 of waves. This corresponds to H_s in Equation 17.99 in the *Fluent Theory Guide*.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Peak Wave Frequency
is the wave frequency corresponding to the highest wave energy. This corresponds to ω_p in Equation 17.99 and Equation 17.100 in the *Fluent Theory Guide*. ω_p can be expressed in terms of the peak wave period, T_p, as $\omega_p = {}^{2\pi}/_{T_p}$.

Minimum/Maximum Wave Frequency
specify the frequency range for the spectrum. Note that the **Peak Wave Frequency** must fall in between **Minimum Wave Frequency** and **Maximum Wave Frequency**.

Number of Frequency Components
specifies the number of components into which the frequency spectrum is divided.

Direction Spreading Method
specifies the method to use for specifying the directional spreading characteristics. You can choose **Uni-directional** (for long-crested waves), **Frequency Independent Cosine Function** (for short-crested waves where the directional component does not depend on frequency), and **Frequency Dependant Hyperbolic Function** (for short-crested waves where the directional component depends on frequency).

Frequency Independent Cosine Exponent
specifies the exponent in the Frequency Independent Cosine Function formulation. This corresponds to s in Equation 17.104 in the *Fluent Theory Guide*.

Mean Wave Heading Angle
specifies the principal wave heading direction.

Angular Spread
specifies the deviation from the mean wave direction for calculating the angular range.

Number of Angular Components
specifies the number of components into which the angular range is divided.

The total number of wave components is the product of **Number of Frequency Components** and **Number of Angular Components**. For modeling random waves, phase differences for the individual wave components are randomly distributed between 0 and 2π.

Note

Any change in the input for **Number of Wave Components** will change the values of phase difference for each component. Therefore, it is recommended that you start the simulation from a saved case file after setting all the wave spectrum user inputs and that you not change the number of wave components during the simulation.

24.3.5.1. Summary Report and Regime Check

A useful text command used to print out a summary of the open channel wave boundary condition settings is `define/boundary-conditions/open-channel-wave-settings`. The output will depend on whether you have chosen **Wave Theories** or **Wave Spectrum**.

Sample Output for Wave Theories

```
/define/boundary-conditions> open-channel-wave-settings

Wave Input Analysis for Velocity Inlet : Thread ID = 5
*******************************************************
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Wave-1 Analysis
**************************************

Current Settings :
------------------
Wave theory : 5th-order-Stokes , Wave regime = Shallow/Intermediate
Wave Height (H) = 0.2160, Wave Length (L) = 1.9400
Liquid Depth (h) = 0.6000, Ursell Number (H*L*L/(h*h*h)) = 3.7636

Mandatory checks for full wave regime within wave breaking limit
-----------------------------------------------------------------
Relative Height: H/h = 0.3600 , Maximum theoretical limit = 0.7800
Maximum numerical limit = 0.5500
Relative height within wave breaking limit

Wave Steepness: H/L = 0.1113 , Maximum theoretical limit = 0.1420
Stable numerical limit = 0.1000 , Maximum numerical limit = 0.1200
Warning: Wave steepness exceeding the stable numerical limit.
Waves could be stable or unstable in this regime.

Checks for selected wave theory within wave breaking and stability limit
-------------------------------------------------------------------------
Relative height check
H/h = 0.3600 , Min : 0.0000 , Max : 0.5000
Relative height check : successful

Wave Steepness check
H/L = 0.1113 , Min : 0.0000 , Max : 0.1363
Wave steepness check : successful

Ursell Number check
Ur = 3.7636 , Min : 0.0000 , Max : 25.0000
Ursell number check : successful

Wave regime check
h/L = 0.3093 , Min : 0.0600 , Max : 10000.0000
Wave regime check : successful

Summary
----------------------
Checks : passed
Selected wave theory is appropriate for application.
```

Sample Output for Wave Spectrum

```
Wave Input Analysis for Velocity Inlet : Thread ID = 6
*******************************************************

Wave Spectrum Analysis
**************************************

Current Settings :
------------------
Frequency Spectrum : Jonswap
Direction Spreading Method : Unidirectional

Significant Wave Height (Hs) = 5.0000
Peak Wave Frequency (wp) = 1.0000, Peak Time Period (Tp) = 6.2832
Minimum Frequency (wi) = 0.6600, Maximum Frequency (we) = 1.6600

Wave Lengths at Min/Peak/Max frequencies (Li, Lp, Le) = (141.5015, 61.6380, 22.3683)

Recommendation: Set the min and max wave frequencies so that most of the wave energy
  is concentrated in the selected regime of spectrum.
  - Min frequency = 0.5*Peak frequency
  - Max frequency = 2.5*Peak frequency


Wave Regime Check
----------------------
Wave regime type = Short Gravity (Deep)
```

```
Information: Validity of short gravity assumption : Liquid Depth > 0.5*Max Wave Length

Sea Steepness Check
-----------------------
Peak Time Period (Tp) = 6.2832
Zero Upcrossing Time Period (Tz) = 4.7869

Sea Steepness based on Tp (Sp) = 0.0811, Steepness Limit = 0.0667
Sea Steepness based on Tz (Sz) = 0.1398, Steepness Limit = 0.1000

Warning: Sea Steepness based on peak time period exceeding limit.
Warning: Sea Steepness based on zero upcrossing time period exceeding limit.

Information: High wave steepness of any individual component too could affect the wave pattern,
  as superposition of wave components is based upon lienar wave theory.

Frequency Spectrum Check
--------------------------
Peak Time Period to Wave Ht Sqrt Ratio (r = Tp/sqrt(Hs)) = 2.8099

Message: Selection of Jonswap spectrum is appropriate.

Recommended value of Peak Shape Parameter for r <= 3.6 :  5.0000

Spectrum Resolution check
---------------------------
Number of frequency components = 50
Number of angular components = 1
Total number of wave components = 50

Information: Higher number of wave components would have adverse effect on speed-up
  whereas smaller could affect the accuracy by not resolving the spectrum well.
  Following steps could assist to optimize the number of components
  1. Set number of frequency/angular components in boundary condition panel,
  2. Set TUI command solve> set> open-channel-wave-verbosity to 2 before initialization,
  3. Initialize (It would print Spectrum Energy for each frequency/angle.)
  4. Copy the information to plot Sw vs Omega and S_theta vs theta
  5. Optimize the number of components after repeating steps 1-4
```

24.3.5.2. Transient Profile Support for Wave Inputs

ANSYS Fluent supports transient profiles for all wave inputs using user-defined functions as seen by the example below:

```
/************************************************************************
  UDF for defining transient profile wave inputs
 ************************************************************************/

#include "udf..h"

#define H 0.02 /* wave height for both waves : same */
#define LEN1 3. /* wave length for first wave */
#define LEN2 6. /* wave length for second wave*/
#define G 9.81
#define D 10. /* liquid depth */
#define U 1. /* wave current */
#define X 0. /* inlet point */

DEFINE_TRANSIENT_PROFILE(wave_ht, current_time)
{
 real k1 = 2.*M_PI/LEN1;
 real k2 = 2.*M_PI/LEN2;
 real w1 = sqrt(G*k1*tanh(k1*D)) + k1*U;
 real w2 = sqrt(G*k2*tanh(k2*D)) + k2*U;
 real dk = 0.5*(k1 - k2);
 real dw = 0.5*(w1 - w2);
 return (2.*H*cos(dk*X - dw*current_time));
}
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.3.5.3. Alternative Stokes Wave Theory Variant

Fluent uses Fenton's formulation for Stokes theories by default. For additional information about the current Stokes theories formulation, see Stokes Wave Theories in the *Fluent Theory Guide*. To revert to the old formulation, provided by Skjebreia and Hendrickson, use the following text command:

```
/solve/set/open-channel-wave-options>
set-buffer-layer-ht     set-verbosity           stokes-wave-variants

/solve/set/open-channel-wave-options> stokes-wave-variants
Use Fenton's formulation for Stokes wave theories [yes] no
Activating old formulation (Skjelbreia and Hendrickson) for Stokes wave theories.
```

For additional information on the old formulation, refer to Lin's book [59] (p. 2890).

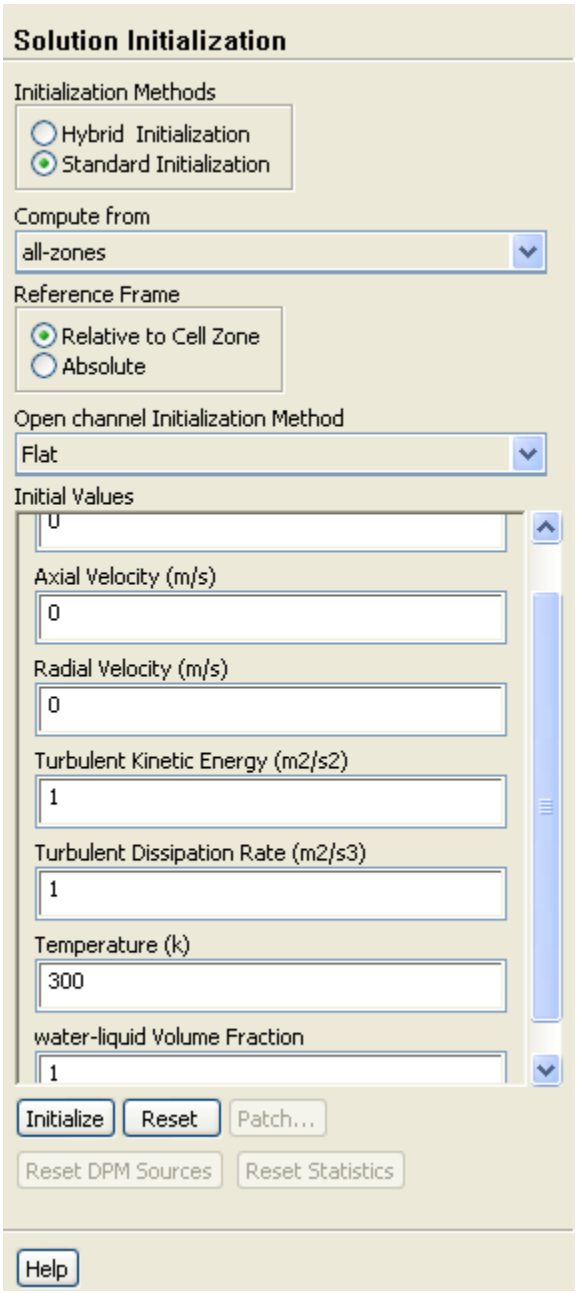
24.3.6. Recommendations for Open Channel Initialization

Once you have selected either the **Open Channel Flow** or the **Open Channel Wave BC** option in the **Multiphase Model** dialog box, then the **Open Channel Initialization Method** drop-down list appears in the **Solution Initialization** task page.

Solution → **Initialization**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.24: The Solution Initialization Task Page



Select an inlet zone from the **Compute from** drop-down list. You can now make your selection from the **Open Channel Initialization Method** drop-down list. If only the **Open Channel Flow** option was enabled, then you only have a choice of **None** or **Flat**. If you enabled **Open Channel Wave BC**, then your choices are **None**, **Flat**, or **Wavy**. The default initialization method is **None**.

If you initialize the solution using **None**, it has no effect as it does not use any open channel information from the selected zone. The **Open Channel Initialization Method** comes into effect when you select either **Flat** or **Wavy**.

Important

This initialization is only valid for pressure-inlets, pressure outlets, and mass-flow inlets for open channel flow and velocity inlets for open channel wave boundary conditions. If the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

selected inlet zone does not have either open channel flow or open channel wave boundary conditions, ANSYS Fluent will report an error message after you initialize the flow with open channel initialization method of **Flat** or **Wavy**.

For open channel initialization from the pressure outlet boundary, the hydrostatic pressure profile based on the **Free Surface Level** is patched in the domain. The volume fraction in the domain is patched based on **Free Surface Level** provided at the pressure outlet boundary. To patch velocity and other variables, the values in the **Solution Initialization** task page will be used.

Important

- Open channel initialization from the pressure outlet boundary is only supported for two phase flow.
- The pressure specification methods, with the exception of **Free Surface Level**, are not supported for open channel initialization from the pressure outlet boundary.

Initialization will result in the volume fraction, X, Y, and Z velocities, and pressure being patched in the domain. The volume fraction will be patched in the domain based on the free surface level of the selected zone from the **Compute from** list. The velocities in the domain will be patched assuming the constant value provided for the velocity magnitude in the selected zone.

Important

If you specify a profile for the velocity magnitude or direction vectors, the initialization will select the value for the velocity magnitude and direction vectors from only one face. Therefore the initialization may be inaccurate. However, generally, open channel inputs for velocity magnitude and direction vectors are constant.

The pressure that is patched is the hydrostatic pressure based on the free surface level specified in the selected zone.

You can use the following text command for open channel automatic initialization:

`solve` → `initialize` → `open-channel-auto-init`

When prompted, set up the following parameters:

boundary thread id
Enter the thread id for the boundary to be selected for open channel automatic initialization.

flat free surface initialization
This option is available for both open channel flow and open channel wave boundary conditions.

wavy free surface initialization
This option appears only for open channel wave boundary conditions, when flat free surface initialization is not selected.

The steps to be followed for open channel automatic initialization are

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Compute defaults based on valid open channel boundary thread. This step is required for better initialization of turbulence parameters based on uniform velocity magnitude.

   `solve → initialize → compute defaults`

2. This would provide information about the selected boundary and type of initialization.

   `solve → initialize → open-channel-auto-init`

3. Initialize

   `solve → initialize → initialize`

24.3.6.1. Reporting Parameters for Open Channel Wave BC Option

To report values as wave speed, wave frequency, and time period for individual waves during initialization, you can set the following text command:

`solve → set → open-channel-wave-options → set-verbosity`

When prompted to set `Verbosity for reporting of derived wave inputs during initialization`, enter `1` as shown below:

Note

You can enter values of 0, 1, or 2. Entering a higher number provides more information.

```
/solve/set/open-channel-wave-options> set-verbosity

Verbosity for reporting of derived wave inputs during initialization
 [0] 1
```

A sample of the resulting output for wave groups after initializing is shown below:

```
Wave-1 : Wave Height = 0.0200, Wave Length = 3.0000
Wave Number = 2.0944, Wave Speed = 2.1642, Wave Frequency = 4.5328
Effective parameters: Wave Speed = 3.1642, Wave Frequency = 6.6272, Time Period = 0.9481

Wave-2 : Wave Height = 0.0200, Wave Length = 6.0000
Wave Number = 1.0472, Wave Speed = 3.0607, Wave Frequency = 3.2052
Effective parameters: Wave Speed = 4.0607, Wave Frequency = 4.2524, Time Period = 1.4776

Time Period based on average effective frequency: 1.1550
```

The resultant output for **Shallow Waves** option is shown below:

```
Wave-1 : Wave Height = 0.2400, Specified Wave Length = 16.0000
Wave Number = 0.5051, Estimated Wave Length = 12.4398
Elliptic Function Parameter (m) : Calculated = 0.9976, Used = 1.0000
Liquid Height = 0.8000, Trough Height = 0.800000
Wave Speed = 3.1868, Wave Frequency = 1.6096
Effective parameters: Wave Speed = 4.1868, Wave Frequency = 2.1147
Time Period based on effective frequency: 2.9712
```

24.3.7. Numerical Beach Treatment for Open Channels

In certain applications, it is desirable to suppress numerical reflection near the outlet boundary for wave dampening. To understand the theory involved in this application, refer to Numerical Beach Treatment.

To include numerical beach in your simulation, perform the following:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Enable **Gravity** and set the gravitational acceleration fields.

   Setup → **General** → **Gravity** → **On**

2. Enable the **Volume of Fluid** model in the **Multiphase Model** dialog box.

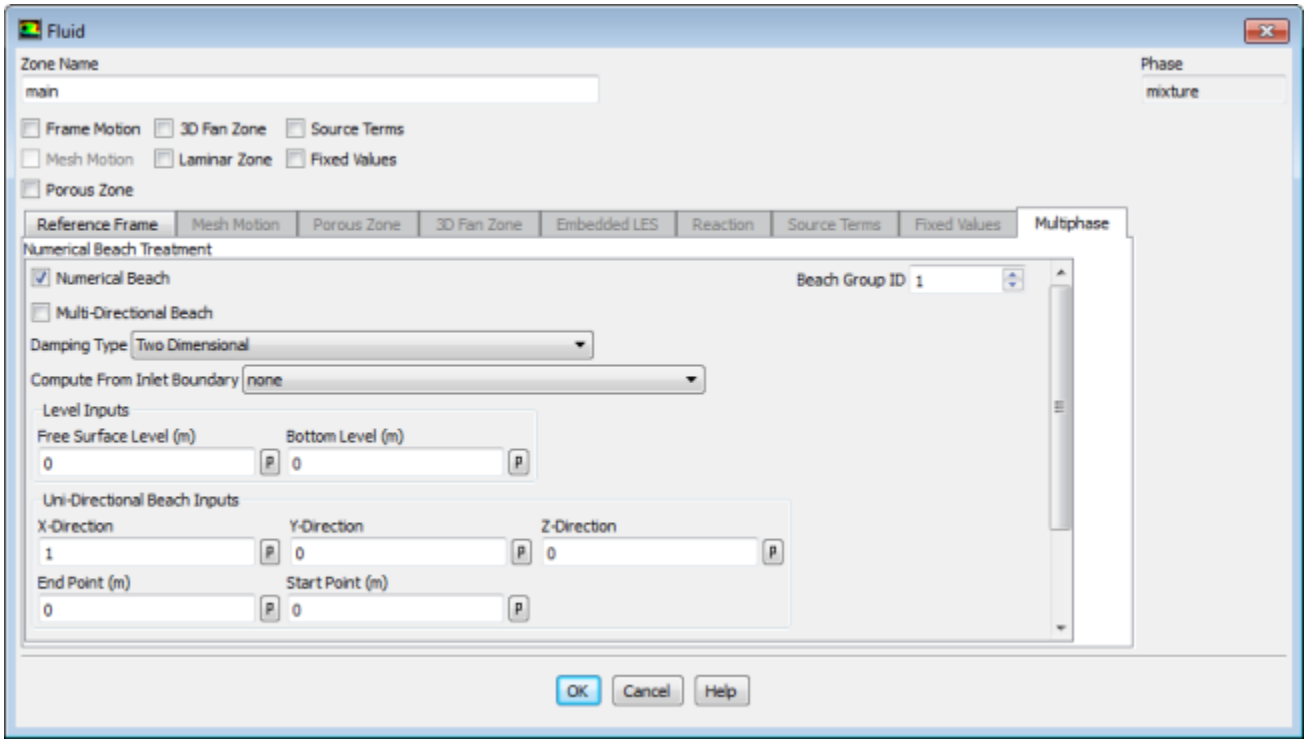
   Setup → **Models** → **Multiphase** → **Edit...**

3. Under **Scheme**, select either **Implicit** or **Explicit**.

4. Select **Open Channel Flow** and/or **Open Channel Wave BC**.

In order to set the numerical beach parameters for a fluid zone, go to the **Fluid** dialog box (Figure 24.25: The Fluid Dialog Box to Enable Numerical Beach (p. 1433)).

Figure 24.25: The Fluid Dialog Box to Enable Numerical Beach



In the **Multiphase** tab of the **Fluid** dialog box, enable the **Numerical Beach** option and enter the following:

- **Beach Group ID** represents the cell zones sharing the damping length containing the same input parameters.
- **Multi-Directional Beach** helps to suppress the numerical reflections that arise from multiple open-channel pressure outlet boundaries. This is desirable while modeling oblique waves or when the pressure-outlet boundaries are closer to the zone of interest. (Only available for 3D problems.)
- **Uni-Directional Beach:** should be used to suppress the numerical reflections from a single open channel pressure outlet. This is the case when the flow is generally uni-directional.
- **Damping Type** allows you to choose between **Two Dimensional** and **One Dimensional**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **Two Dimensional** is the damping treatment in the beach and gravity direction.
  - **One dimensional** is the damping treatment in the beach direction.

- **Compute From Inlet Boundary** is set to **none** by default. If there are available open channel boundaries (velocity-inlet, pressure-inlet, and mass-flow-inlet), boundary names are added to the drop-down list. If you select a boundary from the list, the **Level Inputs**, **Uni-Directional Beach Inputs** or **Multi-Directional Beach Inputs** in beach direction, and **Damping Resistance** values will be updated in the interface. You have the option to overwrite the updated inputs with values that are more applicable to your simulation.
- **Level Inputs** is only available for the **Two Dimensional** damping type.
  - **Free Surface Level** is the same definition as for open channel flow, see Modeling Open Channel Flows (p. 1410).
  - **Bottom Level** is the same definition as for open channel flow, see Modeling Open Channel Flows (p. 1410). During the automatic calculation from the velocity inlet boundary for short gravity waves, this parameter is updated under some assumptions, as noted in Solution Strategies (p. 1435). The bottom level is used for calculating the liquid height.

Numerical Beach Inputs

Numerical beach is assumed as a rectangular domain, where its end-plane (outer surface) is an open channel pressure outlet boundary and start-plane is a plane parallel to a pressure outlet. Beach direction is assumed as the direction normal to a pressure outlet. Damping length is the normal distance between the end and start plane.

Uni-Directional Beach Inputs:

X, **Y**, and **Z** are the vector components of the beach outward normal (typically normal to the nearest pressure outlet).

The end point is calculated by taking the dot product of beach direction with any point lying on the end plane. Similarly, the start point is calculated as the dot product of beach direction with any point lying on the start-plane.

Note

The automatically calculated value of the end point is assumed based on the domain extents for all of the cell zones. Enter your own value if required.

Damping Length Specification is only available if **Open Channel Wave BC** is enabled in the **Multiphase Model** dialog box. There are two options you can choose from:

- **End Point and Wave Lengths** (default).
- **End and Start Points** are the limits of the damping zone.

Multi-Directional Beach Inputs

- **Number of Beach**

  The default is 2, the maximum is 3.

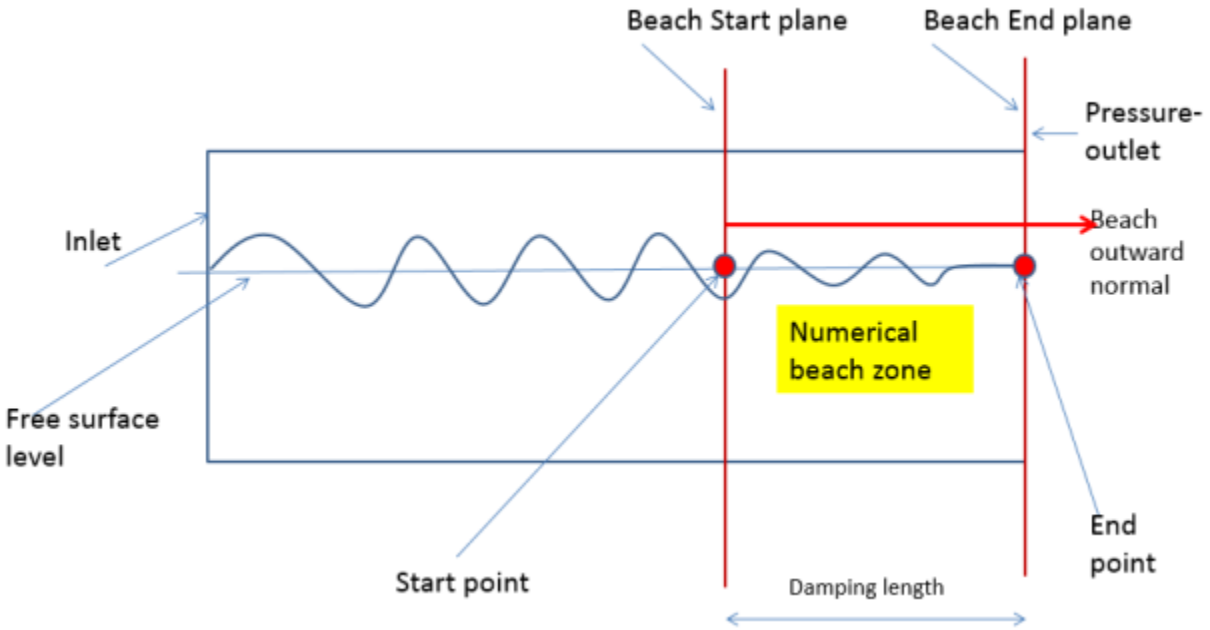
- **X-Direction**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Y-Direction**
- **Z-Direction**
- **End Point**
- **Damping Length**

See Figure 24.26: Numerical Beach Sketch (p. 1435) for a visual representation of the numerical beach.

Figure 24.26: Numerical Beach Sketch



- **Relative Velocity Resistance Formulation** calculates the source term using relative velocities in the numerical beach zone when using moving/deforming meshes or moving reference frames.
- **Linear Damping Resistance** is the resistance per unit time.
- **Quadratic Damping Resistance** is the resistance per unit length.

24.3.7.1. Solution Strategies

Below are some helpful key points when using the **Numerical Beach** option:

1. The **Compute from Inlet Boundary** drop-down list is a convenient feature as it provides automatic inputs, which you should check to make sure the values are reasonable. Some of the provided values are calculated under certain assumptions:

   a. The bottom level, in the case of short gravity waves, is selected in the velocity inlet dialog box for open channel wave boundary conditions. Since there is no user input for this parameter, ANSYS Fluent calculates the bottom level as the value of the free surface level less 2.5 times the wave length. The assumption is that the liquid height is 2.5 times the wave length.

   b. The automatically calculated value of the end point is assumed based on the domain extents for all of the cell zones.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. For manual inputs of start and end points, you should verify that these points are calculated correctly in the beach direction. To check for their validity, subtracting the start point from the end point should give you a positive value.

3. For manual inputs of **Free Surface Level** and **Bottom Level**, you should verify that these points are calculated correctly in the direction opposite to gravity. To check for their validity, subtracting the bottom level from the free surface level should give you a positive value.

4. In case you create separate damping zones, but the damping length is not sufficient to suppress the waves, clubbing of the beach could be done by selecting the same beach ID for other cell zones. In this case, both the cell zones would share the same information.

5. In case of separate damping zones (without clubbing), you should verify that the damping length is more than or equal to the specified number of wave lengths for the calculation of the start point.

6. Damping resistance should be chosen carefully as too much or too little damping could affect the wave profiles in a no-damping zone. The **Compute From Inlet Boundary** option automatically populates the values for damping resistances based on analytical correlations for wave energy. However, you may need to further tune these values for certain cases if the computed values are found to be unsuitable.

7. Steep damping at the beginning of the damping zone could affect the wave profile just before the damping zone.

8. It is recommended that you use a coarse mesh in the damping zone with increased coarseness towards the end of the damping zone.

24.3.8. Defining the Phases for the VOF Model

Instructions for specifying the necessary information for the primary and secondary phases and their interaction in a VOF calculation are provided below.

Important

In general, you can specify the primary and secondary phases whichever way you prefer. It is a good idea, especially in more complicated problems, to consider how your choice will affect the ease of problem setup. For example, if you are planning to patch an initial volume fraction of 1 for one phase in a portion of the domain, it may be more convenient to make that phase a secondary phase. Also, if one of the phases is a compressible ideal gas, it is recommended that you specify it as the primary phase to improve solution stability.

24.3.8.1. Defining the Primary Phase

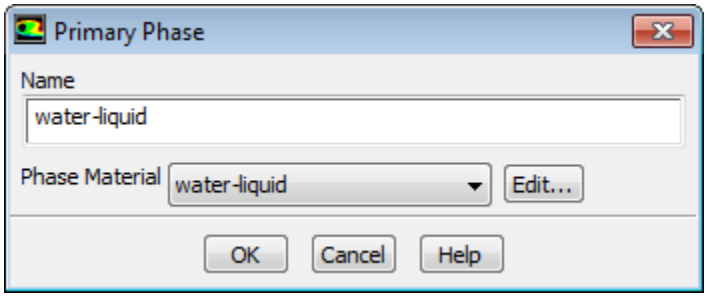
To define the primary phase in a VOF calculation, perform the following steps:

1. Select **phase-1** in the **Phases** list.

2. Click **Edit...** to open the **Primary Phase** dialog box (Figure 24.27: The Primary Phase Dialog Box (p. 1437)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.27: The Primary Phase Dialog Box



3. In the **Primary Phase** dialog box, enter a **Name** for the phase.

4. Specify which material the phase contains by choosing the appropriate material in the **Phase Material** drop-down list.

5. Define the material properties for the **Phase Material**.

   a. Click **Edit...**, and the **Edit Material** dialog box will open.

   b. In the **Edit Material** dialog box, check the properties, and modify them if necessary. (See Physical Properties (p. 459) for general information about setting material properties, Modeling Compressible Flows (p. 1375) for specific information related to compressible VOF calculations, and Modeling Solidification/Melting (p. 1447) for specific information related to melting/solidification VOF calculations.)

   Important

   If you make changes to the properties, remember to click **Change** before closing the **Edit Material** dialog box.

6. Click **OK** in the **Primary Phase** dialog box.

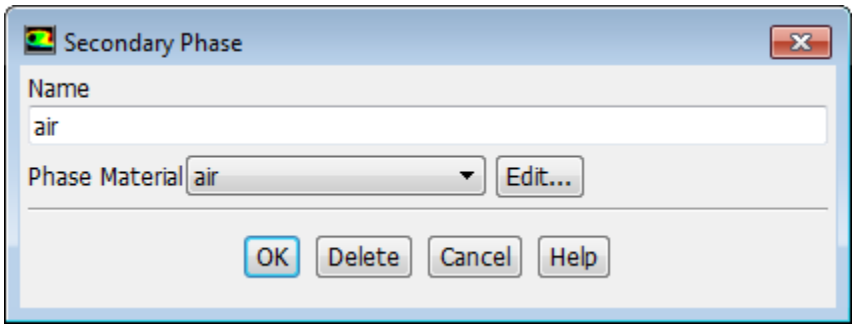
24.3.8.2. Defining a Secondary Phase

To define a secondary phase in a VOF calculation, perform the following steps:

1. Select the phase (for example, **phase-2**) in the **Phases** list.

2. Click **Edit...** to open the **Secondary Phase** dialog box (Figure 24.28: The Secondary Phase Dialog Box for the VOF Model (p. 1437)).

Figure 24.28: The Secondary Phase Dialog Box for the VOF Model



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. In the **Secondary Phase** dialog box, enter a **Name** for the phase.

4. Specify which material the phase contains by choosing the appropriate material in the **Phase Material** drop-down list.

5. Define the material properties for the **Phase Material**, following the procedure outlined above for setting the material properties for the primary phase.

6. Click **OK** in the **Secondary Phase** dialog box.

24.3.8.3. Including Surface Tension and Adhesion Effects

As discussed in When Surface Tension Effects Are Important in the Theory Guide, the importance of surface tension effects depends on the value of the capillary number, Ca (defined by Equation 17.31 in the Theory Guide), or the Weber number, We (defined by Equation 17.32 in the Theory Guide). Surface tension effects can be neglected if $\mathrm{Ca} \gg 1$ or $\mathrm{We} \gg 1$.

Several surface tension options are provided through the text user interface (TUI) using the `solve/set/surface-tension` command :

`solve` → `set` → `surface-tension`

The `surface-tension` command prompts you for the following information:

- whether you require node-based smoothing

  The default value is `yes`, indicating that node-based smoothing will be used. Note that if you are reading in a case that was created in versions prior to ANSYS Fluent 13, then cell-based smoothing will be used by default for the VOF calculations.

- the number of smoothings

  The default value is `1`. A higher value can be used in case of tetrahedral and triangular meshes in order to reduce any spurious velocities.

- the smoothing relaxation factor

  The default is `1`. This is useful in the cases where VOF smoothing causes a problem (for example, liquid enters through the inlet with wall adhesion on).

- whether you want to use VOF gradients at the nodes for curvature calculations

  With this option, ANSYS Fluent uses VOF gradients directly from the nodes to calculate the curvature for surface tension forces. The default is `yes` which produces better results with surface tension compared to gradients that are calculated at the cell centers.

Important

Note that the calculation of surface tension effects will be more accurate if you use a quadrilateral or hexahedral mesh in the area(s) of the computational domain where surface tension is significant. If you cannot use a quadrilateral or hexahedral mesh for the entire domain, then you should use a hybrid mesh, with quadrilaterals or hexahedra in the affected areas.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

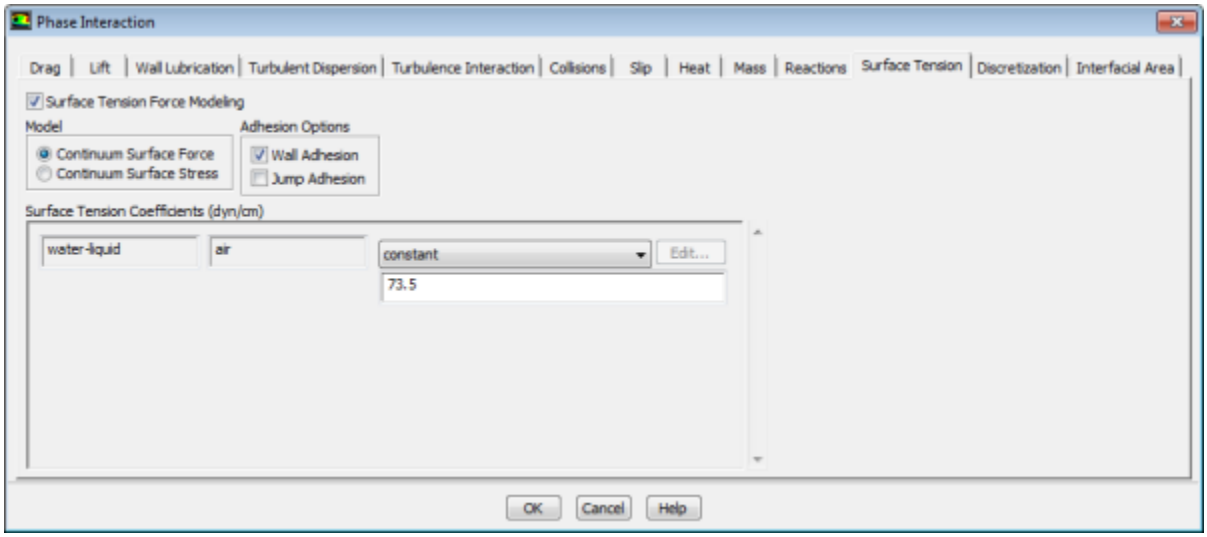
ANSYS Fluent uses node based smoothing for smoothed volume fraction and node based gradients for smoothed curvature calculation to provide better accuracy and robustness.

Important

Pressure jump caused by the surface tension is discontinuous in nature, and also it acts locally at the interface. These numerical difficulties are overcome in an approximate manner by considering the smoothed distribution of the volume fraction field within the finite interfacial width. Smoothing procedures, in general, are mesh dependent. Therefore, the amount and nature of the smoothing procedure could have a significant effect on the results for surface tension cases.

If you want to include the effects of surface tension along the interface between one or more pairs of phases, as described in Surface Tension and Adhesion in the Theory Guide, click **Interaction...** to open the Phase Interaction Dialog Box (p. 2363) (Figure 24.29: The Phase Interaction Dialog Box for the VOF Model (Surface Tension Tab) (p. 1439)).

Figure 24.29: The Phase Interaction Dialog Box for the VOF Model (Surface Tension Tab)



Perform the following steps to model surface tension (and, if appropriate, include adhesion) effects along the interface between one or more pairs of phases:

1. Click the **Surface Tension** tab.

2. Enable the **Surface Tension Force Modeling** option to include the surface tension method.

   Note

   Make sure you specify the **Surface Tension Coefficients** as **constant** or **user-defined**. If **none** is selected, then **Surface Tension Force Modeling** will automatically be disabled.

3. Select the surface tension method that is most applicable to your case. You can choose between **Continuum Surface Force** and **Continuum Surface Stress**. Information about each of the methods is described in Surface Tension in the Theory Guide.

4. For each pair of phases between which you want to include the effects of surface tension, specify an input for the surface tension coefficients using one of the following options:

   - Constant Value
   - Temperature-dependent polynomial, piecewise polynomial, or piecewise linear

     Note

     These options are available while modeling energy.

   - User-defined surface tension coefficient

     Note

     This can be defined as a function of any space variable or time.

   See Surface Tension and Adhesion in the Theory Guide for more information on surface tension, and the Fluent Customization Manual for more information on user-defined functions. All surface tension coefficients are equal to 0 by default, representing no surface tension effects along the interface between the two phases.

   Important

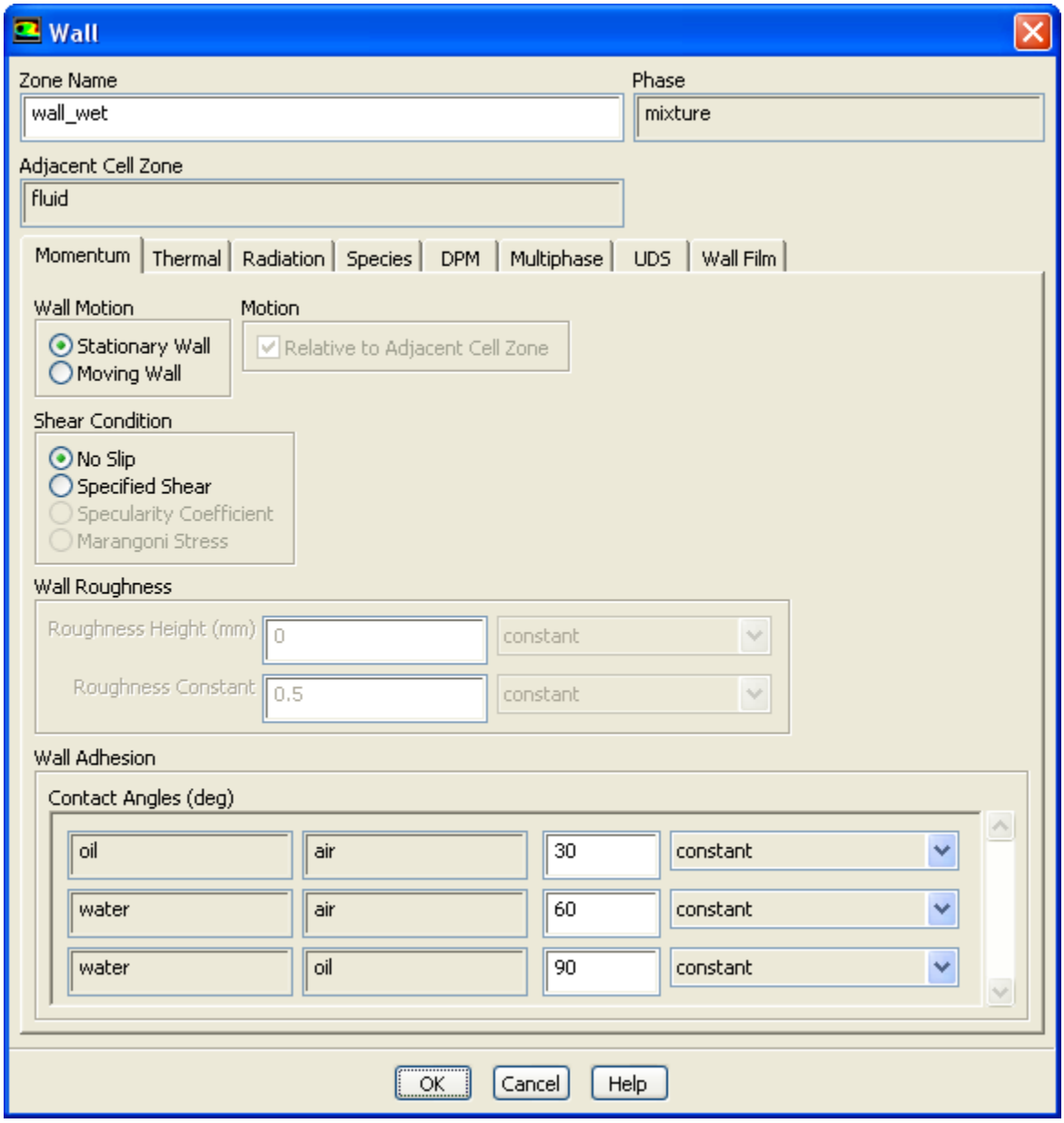
   For calculations involving surface tension, it is recommended that you also turn on the **Implicit Body Force** treatment for the **Body Force Formulation** in the **Multiphase Model** dialog box. This treatment improves solution convergence by accounting for the partial equilibrium of the pressure gradient and surface tension forces in the momentum equations. See Including Body Forces (p. 1377) for details.

5. If you want to include wall adhesion, enable the **Wall Adhesion** option. When **Wall Adhesion** is enabled, you will need to specify the contact angle at each wall as a boundary condition (as described in Defining Multiphase Cell Zone and Boundary Conditions (p. 1391)).

   The contact angle θ_w is the angle between the wall and the tangent to the interface at the wall, measured inside the *phase listed in the left column* under **Wall Adhesion** in the **Momentum** tab of the **Wall** dialog box. For example, if you are setting the contact angle between the oil and air phases in the **Wall** dialog box shown in Figure 24.30: The Wall Dialog Box for a Mixture in a VOF Calculation with Wall Adhesion (p. 1441), θ_w is measured inside the oil phase, as seen in Figure 24.31: Measuring the Contact Angle (p. 1442). For more information, refer to Wall Adhesion in the Theory Guide.

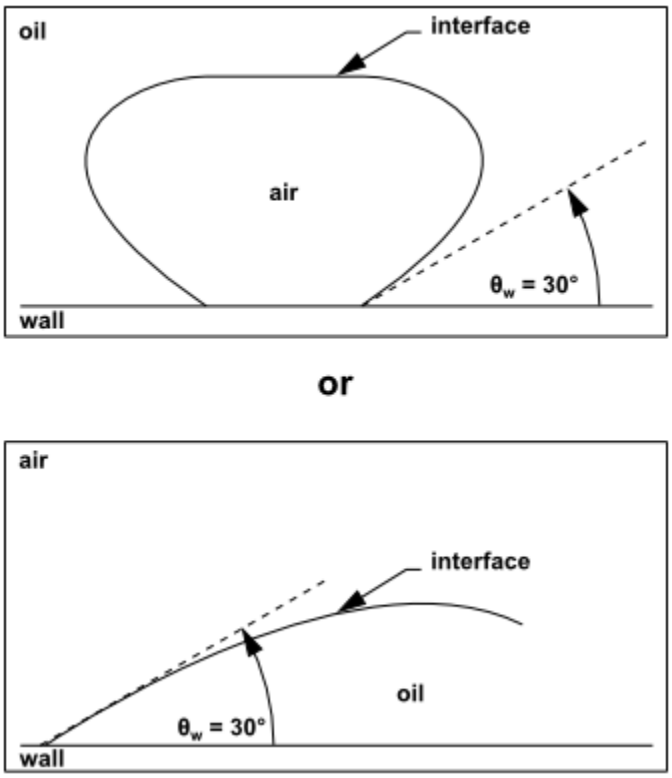
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.30: The Wall Dialog Box for a Mixture in a VOF Calculation with Wall Adhesion



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.31: Measuring the Contact Angle



6. If you want to include jump adhesion, enable the **Jump Adhesion** option. When **Jump Adhesion** is enabled, you will need to specify the contact angle at the porous jump in the **Porous Jump** dialog box (as described in Defining Multiphase Cell Zone and Boundary Conditions (p. 1391)). The contact angle specification and measurement for jump adhesion is similar to that of wall adhesion (as explained in step 3 above).

Important

Jump adhesion only supports the cell based smoothing option for surface tension. When the **Jump Adhesion** option is enabled in the **Phase Interaction** dialog box, the correct settings are automatically set for surface tension. To view the surface tension settings, use the `solve/set/surface-tension` text command.

Note

The following limitations exist:

- When the **Wall Adhesion** and **Jump Adhesion** options are enabled, you must specify a nonzero surface tension coefficient.
- **Jump Adhesion** is not available with the **Continuum Surface Stress** model.
- The **Continuum Surface Stress** model and the **Jump Adhesion** option are not available with the **Level Set** option.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.3.8.4. Discretizing Using the Phase Localized Compressive Scheme

Most of the interface capturing or tracking schemes, which are common in literature, can simulate either diffused interface modeling *or* sharp interface modeling. There are a number of applications where you may be interested in modeling diffused *and* sharp interfaces in different regimes, which results in the need for a unique discretization procedure to handle such applications.

Below are some examples of such applications:

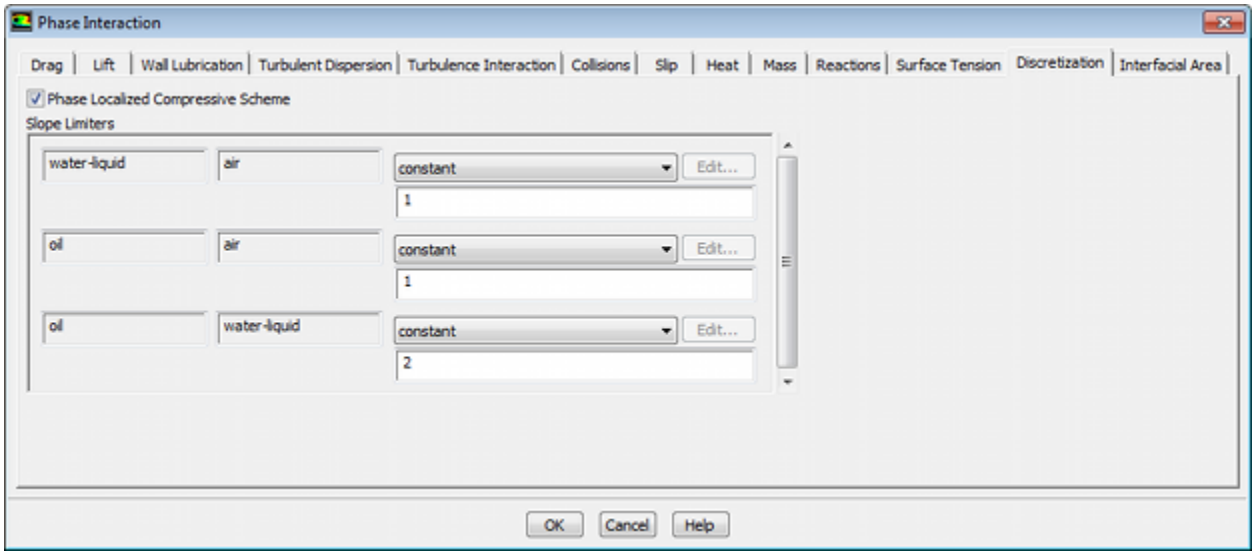
- **Air bubble rise in water-solid slurry:** Diffused interface modeling is required for the water and solid, and sharp interface modeling for the air bubble is desirable. A sharpening scheme would cause undesirable effects for the modeling of the diffused interface between water and solids and a diffusive scheme would not be able to maintain the sharp interface for bubble rising in the slurry.
- **Evaporation/condensation in a tank partially filled with water and being heated at the bottom:** Diffused interface modeling is required for the liquid-vapor and sharp interface modeling for the water-air regime is desirable. A sharpening scheme, would not be able to simulate the diffused phenomena of evaporation/condensation for the liquid- vapor phase and a diffusive scheme would not be able to maintain the sharp interface between water and air.
- **Air jet penetrating through layer of liquids:** You may be interested in diffused jet modeling and sharp interface modeling between the liquid layers. A sharpening scheme would not be able to maintain the continuous air stream and a diffusive scheme would not be able to maintain the sharp interfaces between the layers of liquids.

Using the Modified HRIC and Compressive schemes for such applications would result in the HRIC scheme producing undesirable sharpening of the dispersed phases and undesirable diffusion of the continuous phases, whereas the Compressive scheme would produce undesirable sharpening of the dispersed phases.

Therefore, the phase localized compressive scheme is particularly useful in cases where the desirable behavior is such that you have diffused modeling of dispersed phases and sharp modeling of continuous phases. In ANSYS Fluent, diffusive and anti-diffusive discretization procedures can be used across the distinct interfaces, which share a pair of phases. This functionality is provided through the compressive discretization scheme, where the degree of diffusion or sharpness is controlled through the value of the slope limiters. The theory used is described in The Compressive Scheme and Interface-Model-based Variants.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.32: The Phase Interaction Dialog Box for the VOF Model (Discretization Tab)



To use the phase localized compressive scheme, perform the following steps:

1. Click the **Discretization** tab.

   Note

   The **Discretization** tab is available if the VOF model or the Eulerian multiphase model with **Mulit-Fluid VOF** is enabled, and the **Phase Localized Discretization** option is enabled in the **Interface Modeling Options** dialog box. (Interface Modeling Type (p. 1371)).

2. Enable **Phase Localized Compressive Scheme**.

   Important

   When this option is enabled, the **Compressive** spatial discretization scheme is automatically selected for the **Volume Fraction** in the **Solution Methods** dialog box.

3. For each pair of phases, specify the value of the slope limiter. You can enter a value of 0, 1, or 2, or any value between 0 and 2. Refer to Table 24.11: Slope Limiter Discretization Scheme (p. 1444) to equate each value of the slope limiter with a discretization scheme.

   Table 24.11: Slope Limiter Discretization Scheme

Slope Limiter Value β	Scheme
0	first order upwind
1	second order reconstruction bounded by the global minimum/maximum of the volume fraction
2	compressive

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Slope Limiter Value β	Scheme
$0<\beta<1$ and $1<\beta<2$	blended: where a value between 0 and 1 means blending of the first order and second order and a value between 1 and 2 means blending of the second order and compressive scheme

When using the **Phase Localized Compressive Scheme**, note the following:

- The minimum limit for the slope limiter is 0, whereas the maximum is 2.
- The slope limiter can be interpreted as the degree of compression/anti-diffusion, where 0 demonstrates the minimum compression and 2 demonstrates the maximum compression.
- For interfaces sharing the diffused phases, a slope limiter value of 2 should not be used. A value between 0 and 1 is recommended.
- For interfaces sharing the continuous phases, any value between 0 and 2 can be used depending on the application.
- For interfaces sharing the continuous and diffused phases, any value between 0 and 2 can be used based on the application.
- If you want variable discretization behavior for an interface between two phases, you can do so via the `DEFINE_PROPERTY` user-defined function.

Note

– If the interface has a transition from sharp to diffused modeling, you should not experience any problems.

– If the interface has a transition from diffused to sharp modeling, this transition should be smooth by gradually varying the value of the slope limiter within some transition zone.

24.3.9. Setting Time-Dependent Parameters for the Explicit Volume Fraction Formulation

If you are using the **Explicit** volume fraction formulation in ANSYS Fluent, an explicit solution for the volume fraction is obtained either once each time step or once each iteration, depending upon your inputs to the model. You also have control over the time step used for the volume fraction calculation.

You can access the controls for the volume fraction from the Multiphase Model Dialog Box (p. 2171) by clicking **Expert Options...**

There are two inputs for the time-dependent calculation for the VOF model:

Sub-Time Step Calculation Method

When ANSYS Fluent uses the **Explicit** formulation, the time step used for the volume fraction calculation will not be the same as the time step used for the rest of the transport equations. ANSYS Fluent will refine the time step for VOF automatically, based on your input for the maximum **Courant Number** allowed near the free surface. The Courant number is a dimensionless number that compares the time step in a calculation to the characteristic time of transit of a fluid element across a control volume.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The characteristic transit time for a fluid element across the control volume represents the time taken by the fluid to empty out of the cell. This transit time is taken as the smallest of such time in the region near the fluid interface. A sub time step, for use in the VOF calculations, is computed based on this characteristic time, and the maximum allowed **Courant Number** set in the Multiphase Model Dialog Box (p. 2171). For example, if the maximum allowable **Courant Number** is `0.25` (default), the computed sub time step will have a maximum value equal to a quarter of the minimum transit time for any cell near the interface.

In Fluent, the sub time step size for VOF calculations can be computed using the following options:

- **Velocity Based:** The sub time step is estimated by the cell size and the fluid velocity normal to the interface:

$$\triangle t_v = C\frac{\triangle x}{v_{fluid}} \tag{24.7}$$

  where, C is the Courant Number, v_{fluid} is the fluid velocity, and $\triangle x$ is the cell size.

- **Flux Based**: For this option, the sub time step is estimated based on the cell volume and the summation of outgoing fluxes in the cell:

$$\triangle t_f = C\frac{V}{\sum_{cell} U_f} \tag{24.8}$$

  where, C is the Courant Number, V is the volume, and U_f are the outgoing fluxes.

- **Flux Averaged**: The sub time step size is estimated by volumetric averaging the calculations of the **Flux Based** method for the neighboring cells.

- **Hybrid**: This is the default method for calculating sub time step size. The sub time step size is estimated by blending the **Velocity Based** and **Flux Averaged** calculation methods.

Note

The **Flux Based** calculation method is the most conservative sub-time step size estimation, while the **Velocity Based** calculation is the most aggressive. **Flux Averaged** and **Hybrid** methods fall in between for most cases.

Aggressive sub time step calculation methods generally save time during interface reconstruction. However, the aggressive method can lead to increased number of iterations due to larger transient errors being introduced while integrating over the sub time steps for the interface reconstruction.

Solve VOF Every Iteration

By default, ANSYS Fluent will solve the volume fraction equation(s) once for each time step. This means that the convective flux coefficients appearing in the other transport equations will not be completely updated each iteration, since the volume fraction fields will not change from iteration to iteration.

When ANSYS Fluent solves these equations every iteration, the convective flux coefficients in the other transport equations will be updated based on the updated volume fractions at each iteration.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This choice is the less stable of the two, and requires more computational effort per time step than the default choice.

Important

If you are using sliding meshes, or dynamic meshes with layering and/or remeshing, using the **Solve VOF Every Iteration** option will yield more accurate results, although at a greater computational cost.

24.3.10. Modeling Solidification/Melting

If you are including melting or solidification in your VOF calculation, note the following:

- It is possible to model melting or solidification in a single phase or in multiple phases.
- For phases that are *not* melting or solidifying, you must set the latent heat (L), liquidus temperature ($T_{liquidus}$), and solidus temperature ($T_{solidus}$) to zero.

See Modeling Solidification and Melting (p. 1527) for more information about melting and solidification.

24.4. Setting Up the Mixture Model

For background information about the mixture model and the limitations that apply, refer to Overview in the Theory Guide.

For additional information, see the following sections:
24.4.1. Defining the Phases for the Mixture Model
24.4.2. Including Mixture Drift Force
24.4.3. Including Cavitation Effects

24.4.1. Defining the Phases for the Mixture Model

Instructions for specifying the necessary information for the primary and secondary phases and their interaction for a mixture model calculation are provided below.

24.4.1.1. Defining the Primary Phase

The procedure for defining the primary phase in a mixture model calculation is the same as for a VOF calculation. See Defining the Primary Phase (p. 1436) for details.

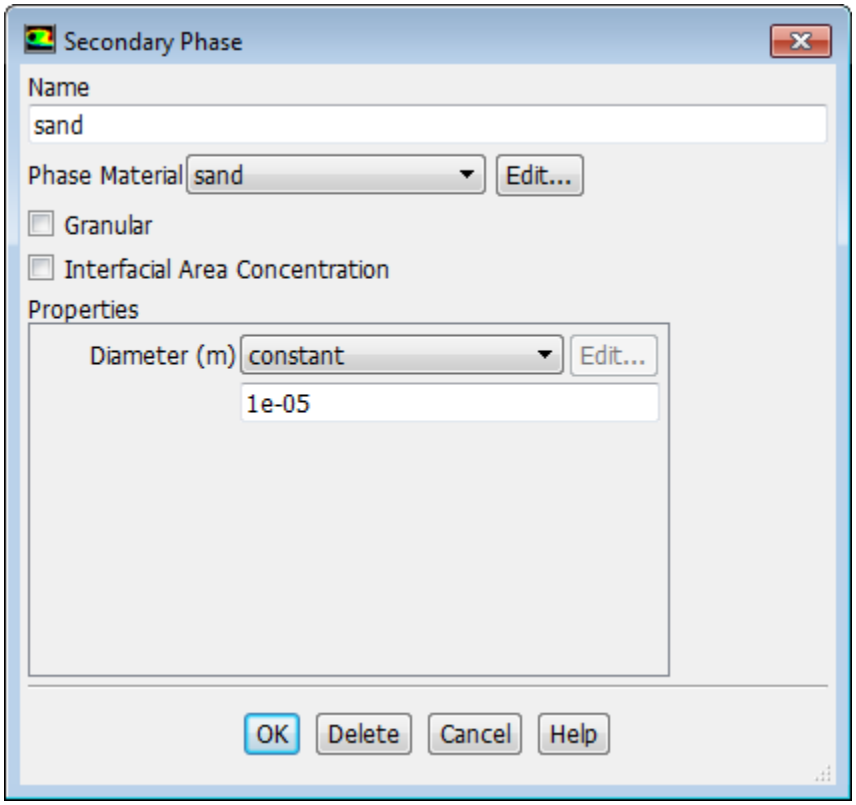
24.4.1.2. Defining a Non-Granular Secondary Phase

To define a non-granular (that is, liquid or vapor) secondary phase in a mixture multiphase calculation, perform the following steps:

1. Select the phase (for example, **phase-2**) in the **Phases** list.
2. Click **Edit...** to open the Secondary Phase Dialog Box (p. 2357) (Figure 24.33: The Secondary Phase Dialog Box for the Mixture Model (p. 1448)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.33: The Secondary Phase Dialog Box for the Mixture Model



3. In the **Secondary Phase** dialog box, enter a **Name** for the phase.

4. Specify which material the phase contains by choosing the appropriate material in the **Phase Material** drop-down list.

5. Define the material properties for the **Phase Material**, following the same procedure you used to set the material properties for the primary phase (see Defining the Primary Phase (p. 1436)). For a particulate phase (which must be placed in the fluid materials category, as mentioned in Steps for Using a Multiphase Model (p. 1365)), you need to specify only the density; you can ignore the values for the other properties, since they will not be used.

6. In the **Secondary Phase** dialog box, specify the **Diameter** of the bubbles, droplets, or particles of this phase (d_p in Equation 17.129 in the Theory Guide). You can specify a constant value, or use a user-defined function. See the Fluent Customization Manual for details about user-defined functions. Note that when you are using the mixture model without slip velocity, this input is not necessary, and it will not be available to you.

7. Click **OK** in the **Secondary Phase** dialog box.

24.4.1.3. Defining a Granular Secondary Phase

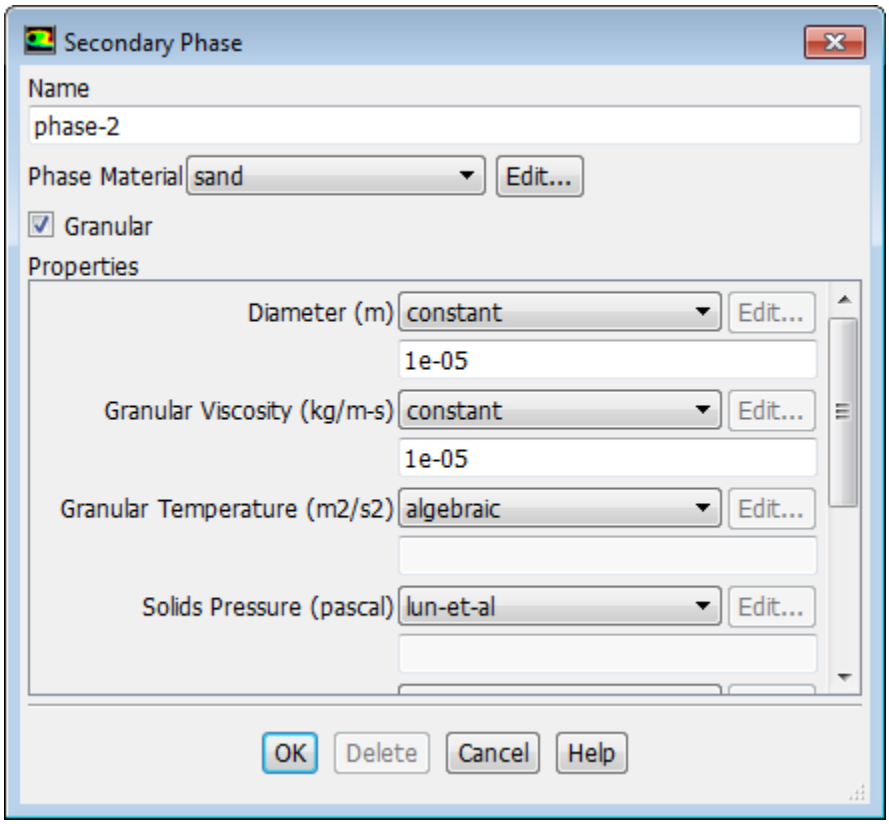
To define a granular (that is, particulate) secondary phase in a mixture model multiphase calculation, perform the following steps:

1. Select the phase (for example, **phase-2**) in the **Phases** list.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click **Edit...** to open the Secondary Phase Dialog Box (p. 2357) (Figure 24.34: The Secondary Phase Dialog Box for a Granular Phase Using the Mixture Model (p. 1449)).

Figure 24.34: The Secondary Phase Dialog Box for a Granular Phase Using the Mixture Model



3. In the **Secondary Phase** dialog box, enter a **Name** for the phase.

4. Specify which material the phase contains by choosing the appropriate material in the **Phase Material** drop-down list.

5. Define the material properties for the **Phase Material**, following the same procedure you used to set the material properties for the primary phase (see Defining the Primary Phase (p. 1436)). For a granular phase (which must be placed in the fluid materials category, as mentioned in Steps for Using a Multiphase Model (p. 1365)), you need to specify only the density; you can ignore the values for the other properties, since they will not be used.

   Important

   Note that all properties for granular flows can be defined by user-defined functions (UDFs).

   See the Fluent Customization Manual for details about user-defined functions.

6. Enable the **Granular** option.

7. In the **Secondary Phase** dialog box, specify the following properties of the particles of this phase:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Diameter

specifies the diameter of the particles. You can select **constant** in the drop-down list and specify a constant value, or select **user-defined** to use a user-defined function. See the Fluent Customization Manual for details about user-defined functions.

Granular Viscosity

specifies the method for computing the kinetic ($\mu_{s,kin}$) and collisional ($\mu_{s,col}$) components of the granular viscosity (Equation 17.137 in the *Fluent Theory Guide*). Selecting **constant**, **syamlal-obrien** (Equation 17.139 in the *Fluent Theory Guide*), or **gidaspow** (Equation 17.140 in the *Fluent Theory Guide*) will use these expressions for the kinetic portion of the viscosity and will calculate the collisional portion of the viscosity from Equation 17.138 in the *Fluent Theory Guide*. Alternatively, you can select **user-defined** to use a user-defined function. Note that if you select **user-defined**, your user-defined function must include both the kinetic portion and the collisional portion of the viscosity in the value it returns.

Frictional Pressure

specifies the pressure gradient term, $\nabla P_{friction}$, in the granular-phase momentum equation. Choose **none** to exclude frictional pressure from your calculation, **johnson-et-al** to apply Equation 17.317 in the Theory Guide, **syamlal-obrien** to apply Equation 17.228 in the Theory Guide, **based-ktgf**, where the frictional pressure is defined by the kinetic theory [26] (p. 2888). The solids pressure tends to a large value near the packing limit, depending on the model selected for the radial distribution function. You must hook a user-defined function when selecting the **user-defined** option. See the Fluent Customization Manual for information on hooking a UDF.

Frictional Modulus

is defined as

$$G=\frac{\partial P_{friction}}{\partial \alpha_{friction}} \tag{24.9}$$

with $G \geq 0$, which is the **derived** option. You can also specify a **user-defined** function for the frictional modulus.

Friction Packing Limit

specifies a threshold volume fraction at which the frictional regime becomes dominant. It is assumed that for a maximum packing limit of 0.6, the frictional regime starts at a volume fraction of about 0.5. This is only a general rule of thumb as there may be other factors involved.

Granular Temperature

specifies temperature for the solids phase and is proportional to the kinetic energy of the random motion of the particles. Choose either the **algebraic**, the **constant**, or **user-defined** option.

Solids Pressure

specifies the pressure gradient term, ∇p_s, in the granular-phase momentum equation. Choose either the **lun-et-al**, the **syamlal-obrien**, the **ma-ahmadi**, or the **user-defined** option.

Radial Distribution

specifies a correction factor that modifies the probability of collisions between grains when the solid granular phase becomes dense. Choose either the **lun-et-al**, the **syamlal-obrien**, the **ma-ahmadi**, the **arastoopour**, or a **user-defined** option.

Elasticity Modulus

is defined as

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$G=\frac{\partial P_s}{\partial \alpha_s} \tag{24.10}$$

with $G \geq 0$.

Choose either the **derived** or **user-defined** options.

Packing Limit

specifies the maximum volume fraction for the granular phase ($\alpha_{s,max}$). For monodispersed spheres, the packing limit is about 0.63, which is the default value in ANSYS Fluent. In polydispersed cases, however, smaller spheres can fill the small gaps between larger spheres, so you may need to increase the maximum packing limit.

8. Click **OK** in the **Secondary Phase** dialog box.

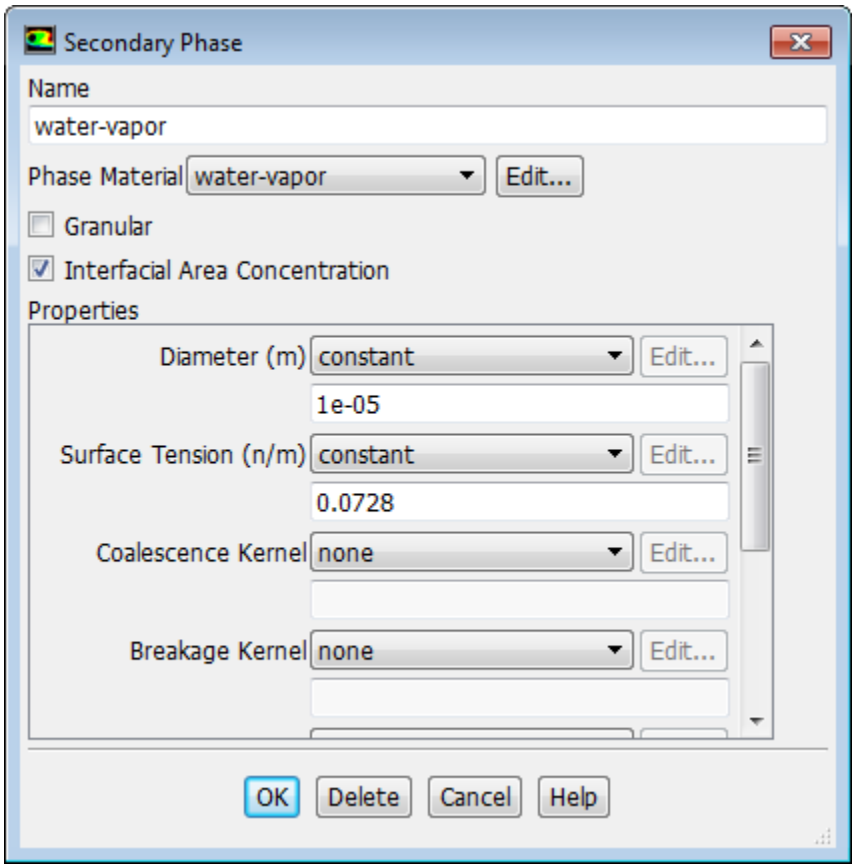
24.4.1.4. Defining the Interfacial Area Concentration

To solve the transport equation for interfacial area concentration of the secondary phase in the mixture model (Interfacial Area Concentration in the *Fluent Theory Guide*, perform the following steps:

1. Select the phase (for example, **phase-2**) in the **Phases** list.

2. Click **Edit...** to open the Secondary Phase Dialog Box (p. 2357) (Figure 24.35: The Secondary Phase Dialog Box Displaying the Interfacial Area Concentration Settings (p. 1451)).

Figure 24.35: The Secondary Phase Dialog Box Displaying the Interfacial Area Concentration Settings



3. In the **Secondary Phase** dialog box, enter a **Name** for the phase.

4. Specify which material the phase contains by choosing the appropriate material in the **Phase Material** drop-down list.

5. Define the material properties for the **Phase Material**.

6. Enable the **Interfacial Area Concentration** option. Make sure the **Granular** option is disabled for the **Interfacial Area Concentration** option to be visible in the interface.

7. In the **Secondary Phase** dialog box, specify the following properties of the particles of this phase:

   Diameter
   specifies the diameter of the particles or bubbles. You can select **constant** in the drop-down list and specify a constant value, or select **user-defined** to use a user-defined function. See the Fluent Customization Manual for details about user-defined functions. The **Diameter** recommended setting is **sauter-mean**, allowing for the effects of the interfacial area concentration values to be considered for mass, momentum and heat transfer across the interface between phases.

   Surface Tension
   specifies the surface tension at the liquid-air interface. You can select either the **hibiki-ishii** or the **ishii-kim** model.

   Coalescence Kernel and Breakage Kernel
   allows you to specify the coalescence and breakage kernels. You can select **none**, **constant**, **hibiki-ishii**, **ishii-kim**, **yao-morel**, or **user-defined**. The three options, **hibiki-ishii**, **ishii-kim** and **yao-morel** are described in detail in Interfacial Area Concentration in the Theory Guide.

   In addition to specifying the **hibiki-ishii**, **ishii-kim**, and **yao-morel** as the coalescence and breakage kernels, you can also tune the properties of the three models by using the `/define/phases/iac-expert/hibiki-ishii-model`, `/define/phases/iac-expert/ishii-kim-model`, and `/define/phases/iac-expert/yao-morel-model` text commands.

   For each of the three models you can specify the parameters listed in Table 24.12: Parameters for the Coalescence and Breakage Kernels (p. 1452)

   Table 24.12: Parameters for the Coalescence and Breakage Kernels

Hibiki-Ishii Model	Ishii-Kim Model	Yao-Morel Model
`Coefficient Gamma_c`	`Coefficient Crc`	`Coefficient K_c1`
`Coefficient K_c`	`Coefficient Cwe`	`Coefficient K_c3`
`Coefficient Gamma_b`	`Coefficient C`	`Coefficient K_b1`
`Coefficient K_b`	`Coefficient Cti`	`alpha_max`
`alpha_max`	`alpha_max`	

   These values are discussed in greater detail in Interfacial Area Concentration in the Theory Guide.

   Nucleation Rate
   is a source term for the interfacial area concentration that models the rate of formation of the dispersed phase. You can choose from **constant** or **user-defined**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Critical Weber Number
will need to be specified if you selected **ishii-kim** or **yao-morel** for the **Breakage Kernel**.

Dissipation Function
gives you the option to choose the formula which calculates the dissipation rate used in the **hibiki-ishii** and **ishii-kim** models. You can choose amongst **constant**, **wu-ishii-kim**, **fluent-ke**, and **user-defined** for the dissipation function.

The **wu-ishii-kim** option uses a simple algebraic correlation for ε:

$$\varepsilon = f_{TW}\left(1/\ 2D_h\right)v_m^3 \tag{24.11}$$

where

$$f_{TW} = \frac{0.316}{\left[(1-\alpha)Re_m\right]^{0.25}}$$

and

$$Re_m = \frac{\rho_m v_m D_h}{\mu_m}$$

where ρ_m, v_m, μ_m, and D_h are the mixture density, mixture velocity, mixture molecular viscosity, and hydraulic diameter of the flow path.

When you select the **wu-ishii-kim** model, you will set an additional input for **Hydraulic Diameter**.

Hydraulic Diameter
is the value used in Equation 24.11 (p. 1453), should you use the **wu-ishii-kim** formulation.

Min/Max Diameter
are the limits of the bubble diameters.

Note

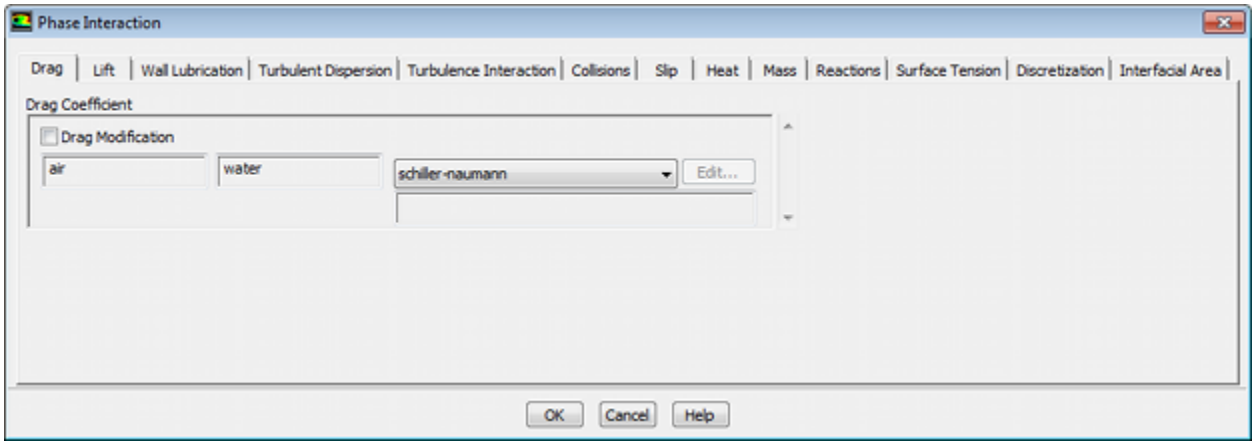
When solving a steady-state problem, the preferred setting for the **Under-Relaxation Factor** is 1.0, as the interfacial area equation for the boiling models is currently under-relaxed using a locally defined pseudo-time step. If you want extra explicit under-relaxation, you may set the value of the **Under-Relaxation Factor** to less than one, this may be done only in case of serious convergence problems with the interfacial area transport equation. To improve convergence you can switch to a pseudo-time step for the interfacial area concentration only, using the **define/phases/iac-expert/iac-pseudo-time-step** text command and set the local pseudo-time to less than 1.

24.4.1.5. Defining Drag Between Phases

For mixture multiphase flows with slip velocity, you can specify the drag function to be used in the calculation. The functions available here are a subset of those discussed in Defining the Phases for the Eulerian Model (p. 1456). See Relative (Slip) Velocity and the Drift Velocity in the Theory Guide for more information.

To specify drag laws, click **Interaction...** to open the Phase Interaction Dialog Box (p. 2363) (Figure 24.36: The Phase Interaction Dialog Box for the Mixture Model (Drag Tab) (p. 1454)), and then click the **Drag** tab.

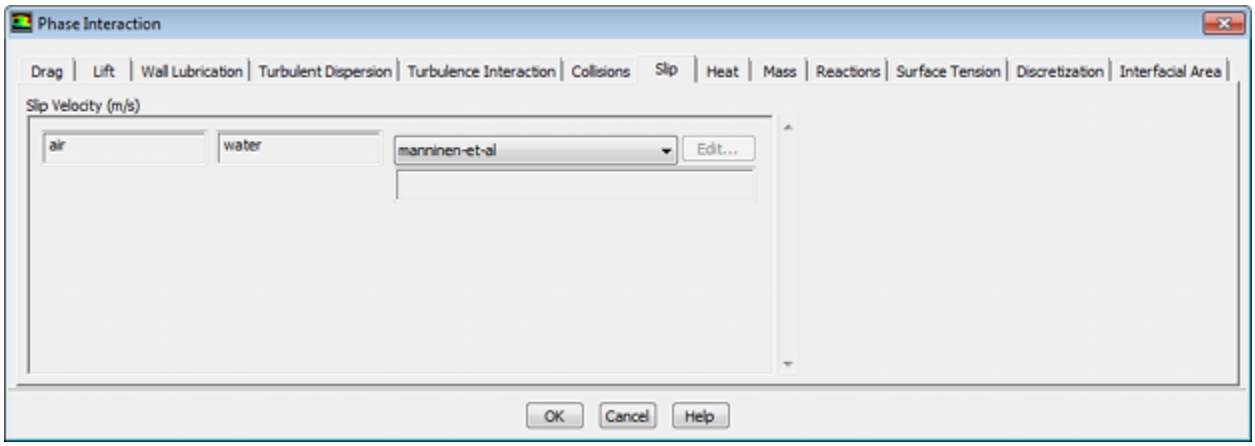
Figure 24.36: The Phase Interaction Dialog Box for the Mixture Model (Drag Tab)



24.4.1.6. Defining the Slip Velocity

If you are solving for slip velocities during the mixture calculation, and you want to modify the slip velocity definition, click **Interaction...** to open the Phase Interaction Dialog Box (p. 2363) (Figure 24.37: The Phase Interaction Dialog Box for the Mixture Model (**Slip** Tab) (p. 1454)), and then click the **Slip** tab.

Figure 24.37: The Phase Interaction Dialog Box for the Mixture Model (Slip Tab)



Under **Slip Velocity**, you can specify the slip velocity function for each secondary phase with respect to the primary phase by choosing the appropriate item in the adjacent drop-down list.

- Select **maninnen-et-al** (the default) to use the algebraic slip method of Manninen et al. [65] (p. 2890), described in Relative (Slip) Velocity and the Drift Velocity in the Theory Guide.
- Select **none** if the secondary phase has the same velocity as the primary phase (that is, no slip velocity).
- Select **user-defined** to use a user-defined function for the slip velocity. See the Fluent Customization Manual for details.

24.4.2. Including Mixture Drift Force

If you are solving for Slip Velocity and using one of the turbulence models in your Mixture multiphase calculation, you can also include the effects of the slip velocity on the momentum and turbulence equations. To include these effects, enable **Mixture Drift Force** in the Viscous Model Dialog Box (p. 2176). Note that the inclusion of these terms can slow down convergence noticeably. If you are looking for

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

additional accuracy, you may want to compute a solution first without these sources, and then continue the calculation with these terms included. In most cases these terms can be neglected.

24.4.3. Including Cavitation Effects

For mixture model calculations, it is possible to include the effects of cavitation, using ANSYS Fluent's cavitation models described in Cavitation Models in the Theory Guide.

To enable the Singhal et al. cavitation model, use the `solve/set/expert` text command and answer `yes` to `use Singhal-et-al cavitation model?`. The **Singhal-Et-Al Cavitation Model** option will now be visible in the **Phase Interaction** dialog box, under the **Mass** tab. Enable this option to include the Singhal et al. cavitation model.

You will specify three parameters to be used in the calculation of mass transfer due to cavitation. Set the **Vaporization Pressure**, the **Surface Tension Coefficient**, and the **Non-Condensable Gas Mass Fraction**. The default value of p_{sat} is 3540 Pa, the vaporization pressure for water at ambient temperature. Note that p_{sat} and the surface tension are properties of the liquid, depending mainly on temperature. **Non-Condensable Gas Mass Fraction** is the mass fraction of dissolved gases, which depends on the purity of the liquid.

When multiple species are included in one or more secondary phases, or the heat transfer due to phase change must be taken into account, the mass transfer mechanism must be defined *before* enabling the cavitation model. It may be noted, however, that for cavitation problems, at least *two* mass transfer mechanisms are defined:

- mass transfer from liquid to vapor.
- mass transfer from vapor to liquid.

To enable and set up the **Schnerr-Sauer** and **Zwart-Gerber-Belamri** cavitation models, refer to Including Mass Transfer Effects (p. 1383).

24.5. Setting Up the Eulerian Model

For background information about the Eulerian model and the limitations that apply, refer to Overview of the Eulerian Model in the Theory Guide.

For additional information, see the following sections:

24.5.1. Additional Guidelines for Eulerian Multiphase Simulations
24.5.2. Defining the Phases for the Eulerian Model
24.5.3. Modeling Turbulence
24.5.4. Including Heat Transfer Effects
24.5.5. Using an Algebraic Interfacial Area Model
24.5.6. Including the Dense Discrete Phase Model
24.5.7. Including the Boiling Model
24.5.8. Including the Multi-Fluid VOF Model

24.5.1. Additional Guidelines for Eulerian Multiphase Simulations

Once you have determined that the Eulerian multiphase model is appropriate for your problem (as described in Choosing a General Multiphase Model in the Theory Guide), you should consider the computational effort required to solve your multiphase problem. The required computational effort depends strongly on the number of transport equations being solved and the degree of coupling. For

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the Eulerian multiphase model, which has a large number of highly coupled transport equations, computational expense will be high. Before setting up your problem, try to reduce the problem statement to the simplest form possible.

Instead of trying to solve your multiphase flow in all of its complexity on your first solution attempt, you can start with simple approximations and work your way up to the final form of the problem definition. Some suggestions for simplifying a multiphase flow problem are listed below:

- Use a hexahedral or quadrilateral mesh (instead of a tetrahedral or triangular mesh).
- Reduce the number of phases.

You may find that even a very simple approximation will provide you with useful information about your problem.

See Eulerian Model (p. 1516) and Setting Solution Limits (p. 1586) for more solution strategies for Eulerian multiphase calculations.

24.5.2. Defining the Phases for the Eulerian Model

Instructions for specifying the necessary information for the primary and secondary phases and their interaction for an Eulerian multiphase calculation are provided below.

24.5.2.1. Defining the Primary Phase

The procedure for defining the primary phase in an Eulerian multiphase calculation is the same as for a VOF calculation. See Defining the Primary Phase (p. 1436) for details.

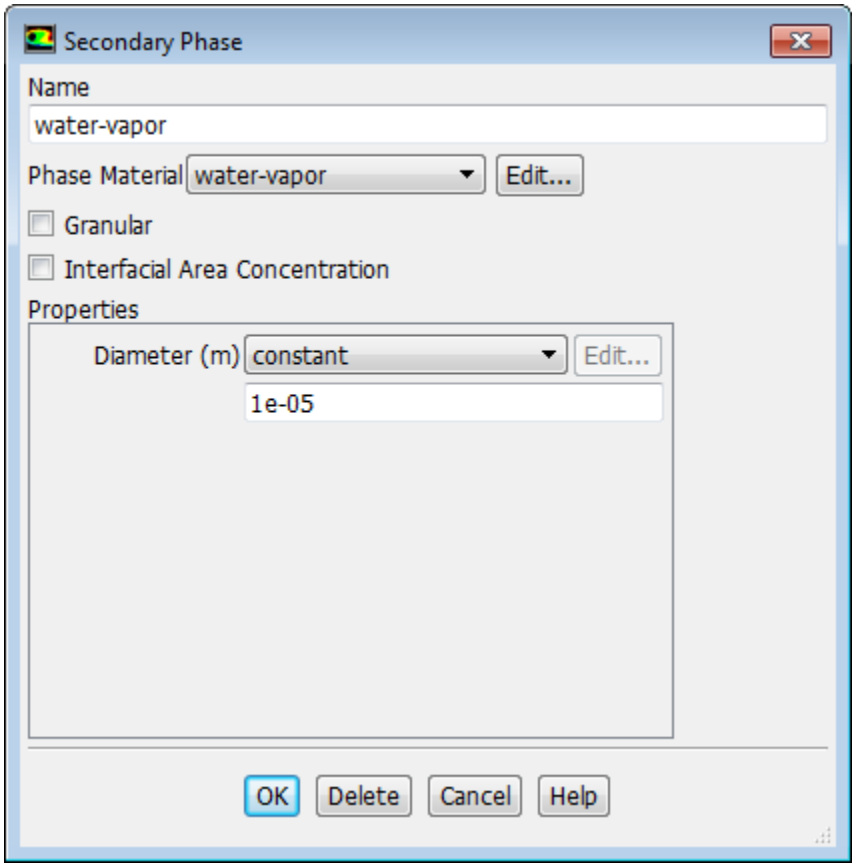
24.5.2.2. Defining a Non-Granular Secondary Phase

To define a non-granular (that is, liquid or vapor) secondary phase in an Eulerian multiphase calculation, perform the following steps:

1. Select the phase (for example, **phase-2**) in the **Phases** list.
2. Click **Edit...** to open the Secondary Phase Dialog Box (p. 2357) (Figure 24.38: The Secondary Phase Dialog Box for a Non-Granular Phase (p. 1457)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.38: The Secondary Phase Dialog Box for a Non-Granular Phase



3. In the **Secondary Phase** dialog box, enter a **Name** for the phase.

4. Specify which material the phase contains by choosing the appropriate material in the **Phase Material** drop-down list.

5. Define the material properties for the **Phase Material**, following the same procedure you used to set the material properties for the primary phase (see Defining the Primary Phase (p. 1436)).

6. In the **Secondary Phase** dialog box, specify the **Diameter** of the bubbles or droplets of this phase. You can specify a constant value, or use a user-defined function. See the Fluent Customization Manual for details about user-defined functions.

7. Click **OK** in the **Secondary Phase** dialog box.

24.5.2.3. Defining a Granular Secondary Phase

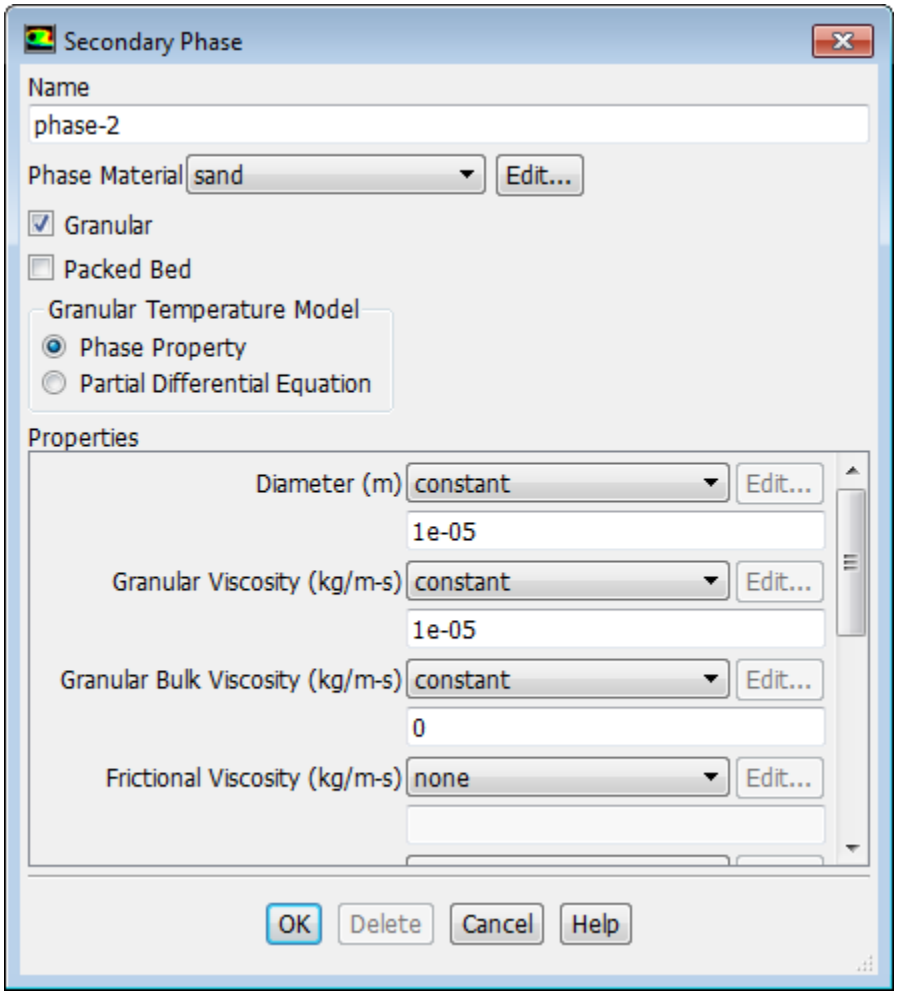
To define a granular (that is, particulate) secondary phase in an Eulerian multiphase calculation, perform the following steps:

1. Select the phase (for example, **phase-2**) in the **Phases** list.

2. Click **Edit...** to open the Secondary Phase Dialog Box (p. 2357) (Figure 24.39: The Secondary Phase Dialog Box for a Granular Phase (p. 1458)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.39: The Secondary Phase Dialog Box for a Granular Phase



3. In the **Secondary Phase** dialog box, enter a **Name** for the phase.

4. Specify which material the phase contains by choosing the appropriate material in the **Phase Material** drop-down list.

5. Define the material properties for the **Phase Material**, following the same procedure you used to set the material properties for the primary phase (see Defining the Primary Phase (p. 1436)). For a granular phase (which must be placed in the fluid materials category, as mentioned in Steps for Using a Multiphase Model (p. 1365)), you need to specify only the density; you can ignore the values for the other properties, since they will not be used.

   Important

   Note that all properties for granular flows can be defined by user-defined functions (UDFs).

   See the Fluent Customization Manual for details about user-defined functions.

6. Enable the **Granular** option.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. (optional) Enable the **Packed Bed** option if you want to freeze the velocity field for the granular phase. Note that when you select the packed bed option for a phase, you should also use the fixed velocity option with a value of zero for all velocity components for all interior cell zones for that phase. Using fixed velocity in radial direction as a constant value (other than zero) does not ensure continuity. With the fixed velocity option, both velocity and pressure are fixed in a zone. Since face area in the radial direction is a function of radial distance from the axis, it will result in a mass conservation problem due to flux imbalance.

   Using zero fixed velocity in the radial direction does not cause any problems related to mass conservation.

8. Specify the **Granular Temperature Model**. Choose either the default **Phase Property** option or the **Partial Differential Equation** option. See Granular Temperature in the Theory Guide for details.

9. In the **Secondary Phase** dialog box, specify the following properties of the particles of this phase:

   Diameter

   specifies the diameter of the particles. You can select **constant** in the drop-down list and specify a constant value, or select **user-defined** to use a user-defined function. See the Fluent Customization Manual for details about user-defined functions.

   Granular Viscosity

   specifies the method for computing the kinetic ($\mu_{s,kin}$) and collisional ($\mu_{s,col}$) components of the granular viscosity (Equation 17.307 in the *Fluent Theory Guide*). Selecting **constant**, **syamlal-obrien** (Equation 17.309), or **gidaspow** (Equation 17.310) will use these expressions for the kinetic portion of the viscosity and will calculate the collisional portion of the viscosity from Equation 17.308 in the *Fluent Theory Guide*. Alternatively, you can select **user-defined** to use a user-defined function. Note that if you select **user-defined**, your user-defined function must include both the kinetic portion and the collisional portion of the viscosity in the value it returns.

   Granular Bulk Viscosity

   specifies the solids bulk viscosity (λ_q in Equation 17.169 in the Theory Guide). You can select **constant** (the default) in the drop-down list and specify a constant value, select **lun-et-al** to compute the value using Equation 17.311 in the Theory Guide, or select **user-defined** to use a user-defined function.

   Frictional Viscosity

   specifies a shear viscosity based on the viscous-plastic flow ($\mu_{s,fr}$ in Equation 17.307 in the Theory Guide). By default, the frictional viscosity is neglected, as indicated by the default selection of **none** in the drop-down list. If you want to include the frictional viscosity, you can select **constant** and specify a constant value, select **schaeffer** to compute the value using Equation 17.312 in the Theory Guide, select **johnson-et-al** to compute the value using Equation 17.317 in the Theory Guide, or select **user-defined** to use a user-defined function.

   Angle of Internal Friction

   specifies a constant value for the angle ϕ used in Schaeffer's expression for frictional viscosity (Equation 17.312 in the Theory Guide). This parameter is relevant only if you have selected **schaeffer** or **user-defined** for the **Frictional Viscosity**.

   Frictional Pressure

   specifies the pressure gradient term, $\nabla P_{friction}$, in the granular-phase momentum equation. Choose **none** to exclude frictional pressure from your calculation, **johnson-et-al** to apply Equation 17.317 in the Theory Guide, **syamlal-obrien** to apply Equation 17.228 in the Theory Guide, **based-ktgf**, where the frictional pressure is defined by the kinetic theory [26] (p. 2888). The solids pressure tends to a large value near the packing limit, depending on the model selected for the radial distribution function. You

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

must hook a user-defined function when selecting the **user-defined** option. See the Fluent Customization Manual for information on hooking a UDF.

Frictional Modulus

is defined as

$$G=\frac{\partial P_{friction}}{\partial \alpha_{friction}} \tag{24.12}$$

with $G \geq 0$, which is the **derived** option. You can also specify a **user-defined** function for the frictional modulus.

Friction Packing Limit

specifies a threshold volume fraction at which the frictional regime becomes dominant. It is assumed that for a maximum packing limit of 0.6, the frictional regime starts at a volume fraction of about 0.5. This is only a general rule of thumb as there may be other factors involved.

Granular Conductivity

specifies the solids granular conductivity (k_{Θ_s} in Equation 17.321 in the Theory Guide). You can select **syamlal-obrien** to compute the value using Equation 17.322 in the Theory Guide, select **gidaspow** to compute the value using Equation 17.323 in the Theory Guide, or select **user-defined** to use a user-defined function. Note that in the algebraic model, shown in Figure 24.39: The Secondary Phase Dialog Box for a Granular Phase (p. 1458), the granular conductivity is not required in the computation of the granular temperature. This has been obtained by neglecting convection and diffusion in the transport equation, Equation 17.321 in the Theory Guide [116] (p. 2893).

Granular Temperature

specifies temperature for the solids phase and is proportional to the kinetic energy of the random motion of the particles. Choose the **algebraic**, **constant**, **dpm-averaged**, or **user-defined** option.

Note

The **dpm-averaged** method is only available when using the Dense Discrete Phase Model (DDPM).

Solids Pressure

specifies the pressure gradient term, ∇p_s, in the granular-phase momentum equation. Choose either the **lun-et-al**, the **syamlal-obrien**, the **ma-ahmadi**, **none**, or a **user-defined** option.

Radial Distribution

specifies a correction factor that modifies the probability of collisions between grains when the solid granular phase becomes dense. Choose either the **lun-et-al**, the **syamlal-obrien**, the **ma-ahmadi**, the **arastoopour**, or a **user-defined** option.

Elasticity Modulus

is defined as

$$G=\frac{\partial P_s}{\partial \alpha_s} \tag{24.13}$$

with $G \geq 0$.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Packing Limit

specifies the maximum volume fraction for the granular phase ($\alpha_{s,max}$). For monodispersed spheres, the packing limit is about 0.63, which is the default value in ANSYS Fluent. In polydispersed cases, however, smaller spheres can fill the small gaps between larger spheres, so you may need to increase the maximum packing limit.

10. Click **OK** in the **Secondary Phase** dialog box.

24.5.2.4. Defining the Interfacial Area Concentration

When using the Eulerian multiphase model, you can choose to solve a transport equation for the interfacial area, which allows for a distribution of secondary phase diameter, or you can use an algebraic model to compute the interfacial area from a specified diameter. To use an algebraic model, refer to Using an Algebraic Interfacial Area Model (p. 1480). To solve the transport equation (Interfacial Area Concentration in the *Fluent Theory Guide*), perform the following steps:

1. Select the phase (for example, **phase-2**) in the **Phases** list.

2. Click **Edit...** to open the Secondary Phase Dialog Box (p. 2357) (Figure 24.35: The Secondary Phase Dialog Box Displaying the Interfacial Area Concentration Settings (p. 1451)).

3. In the **Secondary Phase** dialog box, enter a **Name** for the phase.

4. Specify which material the phase contains by choosing the appropriate material in the **Phase Material** drop-down list.

5. Define the material properties for the **Phase Material**.

6. Enable the **Interfacial Area Concentration** option. Make sure the **Granular** option is disabled for the **Interfacial Area Concentration** option to be visible in the interface.

7. In the **Secondary Phase** dialog box, specify the following properties of the particles of this phase:

   Diameter

   specifies the diameter of the particles or bubbles. You can select **constant** in the drop-down list and specify a constant value, or select **user-defined** to use a user-defined function. See the Fluent Customization Manual for details about user-defined functions. The **Diameter** recommended setting is **sauter-mean**, allowing for the effects of the interfacial area concentration values to be considered for mass, momentum and heat transfer across the interface between phases.

   Coalescence Kernel and Breakage Kernel

   allows you to specify the coalescence and breakage kernels. You can select **none**, **constant**, **hibiki-ishii**, **ishii-kim**, **yao-morel**, or **user-defined**. The three options, **hibiki-ishii**, **ishii-kim** and **yao-morel** are described in detail in Interfacial Area Concentration in the Theory Guide.

   In addition to specifying the **hibiki-ishii**, **ishii-kim**, and **yao-morel** as the coalescence and breakage kernels, you can also tune the properties of the three models by using the `/define/phases/iac-expert/hibiki-ishii-model`, `/define/phases/iac-expert/ishii-kim-model`, and `/define/phases/iac-expert/yao-morel-model` text commands.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For each of the three models you can specify the parameters listed in Table 24.13: Parameters for the Coalescence and Breakage Kernels (p. 1462)

Table 24.13: Parameters for the Coalescence and Breakage Kernels

Hibiki-Ishii Model	Ishii-Kim Model	Yao-Morel Model
Coefficient Gamma_c	Coefficient Crc	Coefficient K_c1
Coefficient K_c	Coefficient Cwe	Coefficient K_c3
Coefficient Gamma_b	Coefficient C	Coefficient K_b1
Coefficient K_b	Coefficient Cti	alpha_max
alpha_max	alpha_max	

These values are discussed in greater detail in Interfacial Area Concentration in the Theory Guide.

Nucleation Rate

is a source term for the interfacial area concentration that models the rate of formation of the dispersed phase. You can choose from **constant** or **user-defined**. If the **Boiling Model** option is enabled, you can also select **yao-morel**. The **yao-morel** option is described in Yao-Morel Model in the Theory Guide.

Critical Weber Number

will need to be specified if you selected **ishii-kim** or **yao-morel** for the **Breakage Kernel**.

Dissipation Function

gives you the option to choose the formula which calculates the dissipation rate used in the **hibiki-ishii** and **ishii-kim** models. You can choose amongst **constant**, **wu-ishii-kim**, **fluent-ke**, and **user-defined** for the dissipation function.

The **wu-ishii-kim** option uses a simple algebraic correlation for ε:

$$\varepsilon = f_{TW}\left(1/\ 2D_h\right)v_m^3 \tag{24.14}$$

where

$$f_{TW} = \frac{0.316}{\left[(1-\alpha)Re_m\right]^{0.25}}$$

and

$$Re_m = \frac{\rho_m v_m D_h}{\mu_m}$$

where ρ_m, v_m, μ_m, and D_h are the mixture density, mixture velocity, mixture molecular viscosity, and hydraulic diameter of the flow path.

When you select the **wu-ishii-kim** model, you will set an additional input for **Hydraulic Diameter**.

Hydraulic Diameter

is the value used in Equation 24.14 (p. 1462), should you use the **wu-ishii-kim** formulation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Min/Max Diameter
allow you to specify the minimum and maximum diameters of the secondary phase to which the interfacial area concentration model is applied, preventing some computing anomalies to spread beyond control.

Note

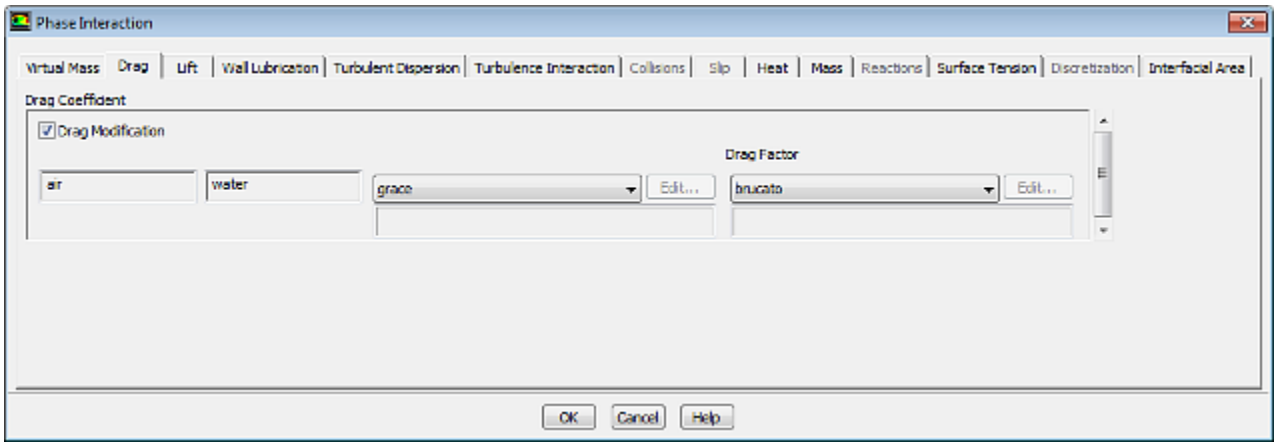
When solving a steady-state problem, the preferred setting for the **Under-Relaxation Factor** is 1.0, as the interfacial area equation for the boiling models is currently under-relaxed using a locally defined pseudo-time step. If you want extra explicit under-relaxation, you may set the value of the **Under-Relaxation Factor** to less than one, this may be done only in case of serious convergence problems with the interfacial area transport equation. To improve convergence you can switch to a pseudo-time step for the interfacial area concentration only, using the **define/phases/iac-expert/iac-pseudo-time-step** text command and set the local pseudo-time to less than 1.

24.5.2.5. Defining the Interaction Between Phases

For both granular and non-granular flows, you will need to specify the drag function to be used in the calculation of the momentum exchange coefficients. You can also specify lift forces, wall lubrication forces (for non-granular flows only), turbulent dispersion forces, surface tension effects, and virtual mass force. For granular flows, you will also need to specify the restitution coefficient(s) for particle collisions.

To specify these parameters, click **Interaction...** to open the Phase Interaction Dialog Box (p. 2363) and visit the **Drag**, **Lift**, **Wall Lubrication**, **Turbulent Dispersion**, **Collisions**, and **Surface Tension** tabs.

Setup → Models → Multiphase → Phases → Edit... → Interaction...



24.5.2.5.1. Specifying the Drag Function

ANSYS Fluent allows you to specify a drag function for each pair of phases. Perform the following steps:

1. Click the **Drag** tab.

2. For each pair of phases, select the appropriate drag function from the corresponding drop-down list.

   - Select **ishii** to use the fluid-fluid drag function described by Equation 17.419 in the Theory Guide. This option is available when the boiling model is enabled.

- Select **schiller-naumann** to use the fluid-fluid drag function described by Equation 17.182 in the Theory Guide. The Schiller and Naumann model is the default method, and it is acceptable for general use in all fluid-fluid multiphase calculations. When using the homogeneous population balance models you can also select **schiller-naumann-pb** in which case the interfacial area will be calculated directly from the population balance variables.

- Select **morsi-alexander** to use the fluid-fluid drag function described by Equation 17.186 in the Theory Guide. The Morsi and Alexander model is the most complete, adjusting the function definition frequently over a large range of Reynolds numbers, but calculations with this model may be less stable than with the other models.

- Select **symmetric** to use the fluid-fluid drag function described by Equation 17.192 in the Theory Guide. The symmetric model is recommended for flows in which the secondary (dispersed) phase in one region of the domain becomes the primary (continuous) phase in another. For example, if air is injected into the bottom of a container filled halfway with water, the air is the dispersed phase in the bottom half of the container; in the top half of the container, the air is the continuous phase. The **symmetric** drag law is the default method for the **Multi-Fluid VOF Model**, which is available with Eulerian multiphase model.

- Select **grace** to use the fluid-fluid drag function described by Equation 17.197 in the Theory Guide. The Grace model is recommended for gas-liquid flows in which the bubbles can have a range of shapes such as spherical, elliptical, or cap.

- Select **tomiyama** to use the fluid-fluid drag function described by Equation 17.203 in the Theory Guide. Like the Grace model, the Tomiyama model is recommended for gas-liquid flows in which the bubbles can have a range of shapes such as spherical, elliptical, or cap.

- Select **anisotropic** to use the fluid-fluid drag function described in Multi-Fluid VOF Model in the Theory Guide. The **anisotropic** drag law is recommended for free surface modeling. It is based on higher drag in the normal direction to the interface and lower drag in the tangential direction to the interface.

- Select **universal-drag** for bubble-liquid and/or droplet-gas flow when the characteristic length of the flow domain is much greater than the averaged size of the droplets/bubbles. It is suitable for flows in which the bubbles/droplets may have a range of shapes. The universal drag law is described using Equation 17.207 in the Theory Guide. When **universal-drag** is selected, you will need to set a value for the surface tension coefficient, under the **Surface Tension** tab, in the **Phase Interaction** dialog box. This value will apply to the primary phase and the secondary phase.

- Select **wen-yu** to use the fluid-solid drag function described by Equation 17.242 in the Theory Guide. The Wen and Yu model is applicable for dilute phase flows, in which the total secondary phase volume fraction is significantly lower than that of the primary phase. When using the homogeneous population balance models you can also select **wen-yu-pb** in which case the interfacial area will be calculated directly from the population balance variables.

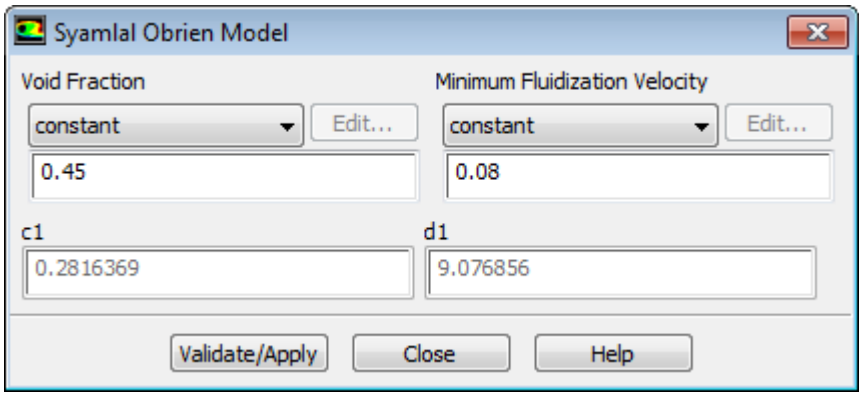
- Select **gidaspow** to use the fluid-solid drag function described by Equation 17.244 in the Theory Guide. The Gidaspow model is recommended for dense fluidized beds.

- Select **syamlal-obrien** to use the fluid-solid drag function described by Equation 17.229 in the Theory Guide. The Syamlal-O'Brien model is recommended for use in conjunction with the Syamlal-O'Brien model for granular viscosity.

- Select **syamlal-obrien-para** to use the parameterized formulation of the Syamlal-O'Brien model described by Equation 17.239 in the Theory Guide. This model addresses under/over-prediction of bed expansion

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

that can arise with **syamlal-obrien**. When you first select this model, you will be presented with the **Syamlal Obrien Model** dialog box.



There are two inputs:

Void Fraction
the volume fraction of the gas phase in the bed at the minimum fluidization condition

Minimum Fluidization Velocity
the expected or experimentally-determined nominal minimum fluidization velocity of the gas phase

Once you have input these values, click **Validate/Apply** to compute the coefficients **c1** and **d1** used in Equation 17.240 in the *Fluent Theory Guide*. A message will also be printed to the text user interface summarizing the fluid and flow properties used in the computation and the resulting values. You should check that these property values are consistent with the properties set elsewhere.

Important

If you change property values in the **Create/Edit Materials** dialog box or diameter in the **Secondary Phase** dialog box, you must initialize the flow field or run at least one iteration before computing c1 and d1. Otherwise, incorrect properties will be used in the computation.

Note that the **syamlal-obrien-para** model is appropriate only for Geldart Group B particles and is only applicable to cases involving a single secondary phase.

- Select **syamlal-obrien-symmetric** to use the solid-solid drag function described by Equation 17.250 in the Theory Guide. The symmetric Syamlal-O'Brien model is appropriate for a pair of solid phases.
- Select **huilin-gidaspow** to use the fluid-solid drag function described by Equation 17.246 in the Theory Guide. This option provides a better blending function for the Gidaspow model when moving from the dense packing limit to the dilute flow limit.
- Select **gibilaro** to use the fluid-solid drag function described by Equation 17.248 in the Theory Guide. This option is used for circulating fluidized beds
- Select **constant** to specify a constant value for the drag function, and then specify the value in the text field.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select **user-defined** to use a user-defined function for the drag function (see the Fluent Customization Manual for details).
- If you want to temporarily ignore the interaction between two phases, select **none**.

24.5.2.5.1.1. Drag Modification

When using the Eulerian or Mixture multiphase models, you can optionally specify a drag modification term for the selected drag law. The drag modification term acts as a multiplier for the drag coefficient computed from the models detailed in Specifying the Drag Function (p. 1463). You can specify the drag modification term individually for each pair of primary-secondary phases. To enable drag modification, perform the following steps:

1. Check **Drag Modification** in the **Drag** tab of the **Phase Interaction** dialog box.

   *An additional drop-down list will appear under **Drag Factor** for each primary-secondary phase pair.*

2. For each pair of phases, select the **Drag Factor** to use from the drop-down list.
   - Select **none** to use the selected drag model without modification.
   - Select **constant** to specify a constant value for the drag modification factor.
   - Select **brucato** to use the Brucato et al. correlation described by Brucato et al. Correlation in the Theory Guide.
   - Select **user-defined** to use a user-defined function for the drag modification factor (see `DEFINE_EXCHANGE_PROPERTY` in the Fluent Customization Manual for details).

For additional details on the implementation of the drag modification, see Drag Modification in the Theory Guide.

24.5.2.5.2. Specifying the Restitution Coefficients (Granular Flow Only)

For granular flows, you need to specify the coefficients of restitution for collisions between particles (e_{ls} in Equation 17.250 and e_{ss} in Equation 17.290 in the Theory Guide). In addition to specifying the restitution coefficient for collisions between each pair of granular phases, you will also specify the restitution coefficient for collisions between particles of the same phase.

Perform the following steps:

1. Click the **Collisions** tab to display the **Restitution Coefficient** inputs.
2. For each pair of phases, specify a constant restitution coefficient. All restitution coefficients are equal to 0.9 by default.

24.5.2.5.3. Including the Lift Force

For both granular and non-granular flows, it is possible to include the effect of lift forces ($\vec{F}_{lift}$ in Equation 17.255 in the Theory Guide) on the secondary phase particles, droplets, or bubbles. These lift forces act on a particle, droplet, or bubble mainly due to velocity gradients in the primary-phase flow field. In most cases, the lift force is insignificant compared to the drag force, so there is no reason to

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

include it. If the lift force is significant (for example, if the phases separate quickly), you may want to include this effect.

Important

Note that the lift force will be more significant for larger particles, but the ANSYS Fluent model assumes that the particle diameter is much smaller than the interparticle spacing. Therefore, the inclusion of lift forces is not appropriate for closely packed particles or for very small particles.

To include the effect of lift forces, perform the following steps:

1. Click the **Lift** tab to display the **Lift Coefficient** inputs.

2. For each pair of phases, select the appropriate specification method from the corresponding drop-down list. Note that, since the lift forces for a particle, droplet, or bubble are due mainly to velocity gradients in the primary-phase flow field, you will not specify lift coefficients for pairs consisting of two secondary phases; lift coefficients are specified only for pairs consisting of a secondary phase and the primary phase.

   - Select **none** (the default) to ignore the effect of lift forces.
   - Select **constant** to specify a constant lift coefficient, and then specify the value in the text field.
   - Select **moraga** to use the Moraga lift model (Moraga Lift Force Model in the Theory Guide). The Moraga lift model is applicable to spherical solid particles, drops, and bubbles.
   - Select **saffman-mei** to use the Saffman-Mei lift model (Saffman-Mei Lift Force Model in the Theory Guide). The Saffman-Mei lift model is applicable to spherical solid particles, and to drops and bubbles that are not significantly distorted.
   - Select **legendre-magnaudet** to use the Legendre—Magnaudet lift model (Legendre-Magnaudet Lift Force Model in the Theory Guide). The Legendre-Magnaudet model is applicable to small diameter spherical fluid particles, though it can be applied to non-distorted liquid drops and bubbles. It accounts for momentum transfer between the flow around the particle and the inner recirculation flow inside the fluid particle caused by fluid friction/stresses at the fluid interface.
   - Select **tomiyama** to use the Tomiyama lift model (Tomiyama Lift Force Model in the Theory Guide). The Tomiyama model is applicable to larger-scale deformable bubbles in the ellipsoidal and spherical cap regimes. Its main feature is the prediction of the cross-over point in bubble size at which particle distortion causes a reversal in the sign of the lift force.
   - Select **user-defined** to use a user-defined function for the lift coefficient (see the Fluent Customization Manual for details).

24.5.2.5.4. Including the Wall Lubrication Force

For liquid-gas bubbly flows using the Eulerian multiphase model, you can include the effect of wall lubrication forces ($\vec{F}_{wl}$ in Equation 17.168 in the Theory Guide) on the secondary phase bubbles. These forces tend to push the bubbles away from walls at small wall distances. For details on how wall lubrication is modeled in ANSYS Fluent see Wall Lubrication Force in the Theory Guide.

To include the effect of wall lubrication forces, perform the following steps:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

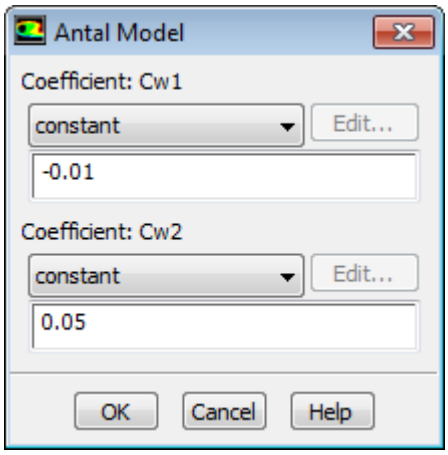
1. Click the **Wall Lubrication** tab to display the **Wall Lubrication** inputs.

2. For each pair of phases, select the appropriate specification method from the corresponding drop-down list. Note that you will specify the wall lubrication only for pairs consisting of a secondary phase and the primary phase.

   - Select **none** (the default) to ignore the effect of wall lubrication forces.

   - Select **antal-et-al** to use the Antal et al. model (Antal et al. Model in the Theory Guide). You can edit the model parameters in the **Antal Model** dialog box (Figure 24.40: Antal et al. Model Dialog Box (p. 1468)).

     Figure 24.40: Antal et al. Model Dialog Box

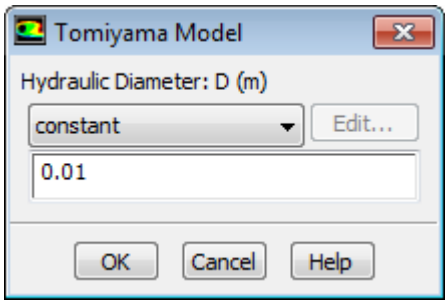
     The following model parameters are available:

     Coefficient: Cw1
     Specify the constant C_{w1} in Equation 17.269 in the Theory Guide. You can enter a constant value or specify a user-defined function defined using the `DEFINE_EXCHANGE_PROPERTY` macro (`DEFINE_EXCHANGE_PROPERTY`).

     Coefficient: Cw2
     Specify the constant C_{w2} in Equation 17.269 in the Theory Guide. You can enter a constant value or specify a user-defined function defined using the `DEFINE_EXCHANGE_PROPERTY` macro (`DEFINE_EXCHANGE_PROPERTY`).

   - Select **tomiyama** to use the Tomiyama model (Tomiyama Model in the Theory Guide). This model is only applicable for pipe geometries. You can enter the Hydraulic Diameter for your geometry in the **Tomiyama Model** dialog box (Figure 24.41: Tomiyama Model Dialog Box (p. 1468)).

     Figure 24.41: Tomiyama Model Dialog Box

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

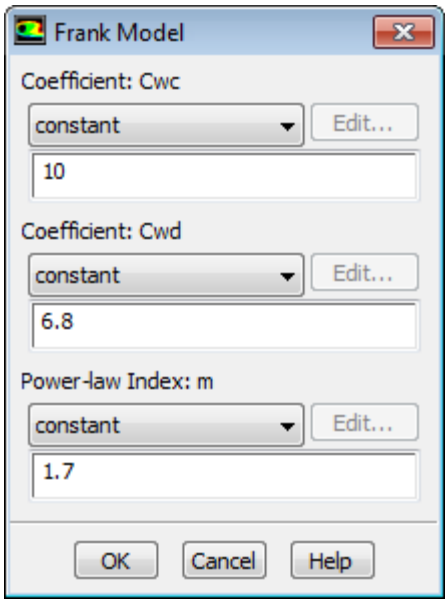
The following model parameters are available:

Hydraulic Diameter: D (m)
Specify the hydraulic diameter, D, in Equation 17.271 in the Theory Guide.

> **Important**
>
> The hydraulic diameter must be entered in units of meters.

- Select **frank** to use the Frank et al. model (Frank Model in the Theory Guide). You can edit the model parameters in the **Frank Model** dialog box (Figure 24.42: Frank Model Dialog Box (p. 1469)).

Figure 24.42: Frank Model Dialog Box



The following model parameters are available:

Coefficient: Cwc
Specify the cutoff coefficient, C_{wc}, in Equation 17.273 in the Theory Guide. You can enter a constant value or specify a user-defined function defined using the DEFINE_EXCHANGE_PROPERTY macro (DEFINE_EXCHANGE_PROPERTY).

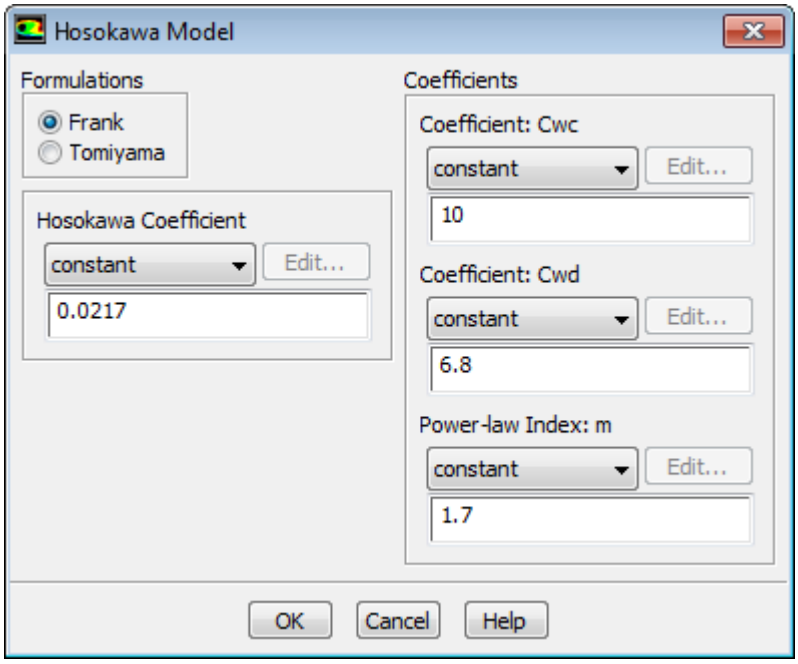
Coefficient: Cwd
Specify the damping coefficient, C_{wd}, in Equation 17.273 in the Theory Guide. You can enter a constant value or specify a user-defined function defined using the DEFINE_EXCHANGE_PROPERTY macro (DEFINE_EXCHANGE_PROPERTY).

Power-law Index: m
Specify the power law constant, m, in Equation 17.273 in the Theory Guide. You can enter a constant value or specify a user-defined function defined using the DEFINE_EXCHANGE_PROPERTY macro (DEFINE_EXCHANGE_PROPERTY).

- Select **hosakawa** to use the Hosokawa model (Hosokawa Model in the Theory Guide). You can edit the model parameters in the **Hosokawa Model** dialog box (Figure 24.43: Hosokawa Model Dialog Box (p. 1470)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.43: Hosokawa Model Dialog Box



The following model parameters are available:

Formulations
Select which formulation of the Hosokawa model to use. You can select either **Frank** or **Tomiyama**.

Hosokawa Coefficient
Specify the coefficient of the Eotvos number in Equation 17.274 in the Theory Guide. You can enter a constant value or specify a user-defined function defined using the DEFINE_EXCHANGE_PROPERTY macro (DEFINE_EXCHANGE_PROPERTY).

Coefficients
Specify the model parameters associated with the formulation you selected under **Formulations**.

- Select **user-defined** to use a user-defined function for the wall lubrication coefficient (see DEFINE_EXCHANGE_PROPERTY in the *Fluent Customization Manual* for details).

24.5.2.5.5. Including the Turbulent Dispersion Force

For turbulent flows using the Eulerian multiphase model, you can include the effects of turbulent dispersion force ($\vec{F}_{td}$ in Equation 17.168 in the Theory Guide). The turbulent dispersion force acts as a turbulent diffusion in dispersed flows. For details on how turbulent dispersion is modeled in ANSYS Fluent see Turbulent Dispersion Force in the Theory Guide.

To include the effects of turbulent dispersion force, perform the following steps:

1. Click the **Turbulent Dispersion** tab to display the **Turbulent Dispersion** inputs.

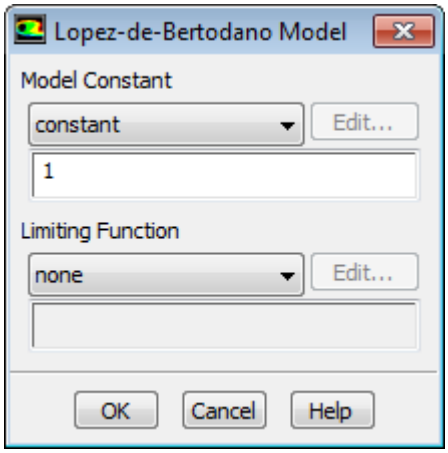
2. For each pair of phases, select the appropriate specification method from the corresponding drop-down list. Note that you will specify the turbulent dispersion only for pairs consisting of a secondary phase and the primary phase.

   - Select **none** (the default) to ignore the effects of turbulent dispersion.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select **lopez-de-bertodano** to use the Lopez de Bertodano model (Lopez de Bertodano Model in the Theory Guide). You can edit the model parameters in the **Lopez de Bertodano Model** dialog box (Figure 24.44: Lopez de Bertodano Model Dialog Box (p. 1471)).

Figure 24.44: Lopez de Bertodano Model Dialog Box



The following model parameters are available:

Model Constant

Specify the coefficient, C_{TD}, in Equation 17.278 in the Theory Guide. You can enter a constant value or specify a user-defined function defined using the DEFINE_EXCHANGE_PROPERTY macro (DEFINE_EXCHANGE_PROPERTY).

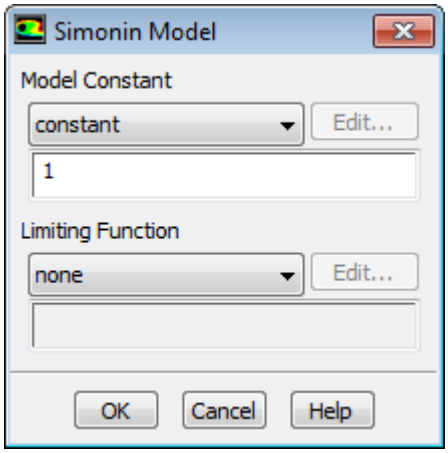
Limiting Function

Specify the limiting function to use. You may select **none**, the **standard** limiting function, or a **user-defined** function using the DEFINE_EXCHANGE_PROPERTY macro (DEFINE_EXCHANGE_PROPERTY). See Limiting Functions for the Turbulent Dispersion Force in Fluent Theory Guide for details on the limiting functions.

- Select **simonin** to use the Simonin model (Simonin Model in the Theory Guide).

You can specify the model parameters in the **Simonin Model** dialog box (Figure 24.45: Simonin Model Dialog Box (p. 1471)).

Figure 24.45: Simonin Model Dialog Box



The following model parameters are available:

Model Constant

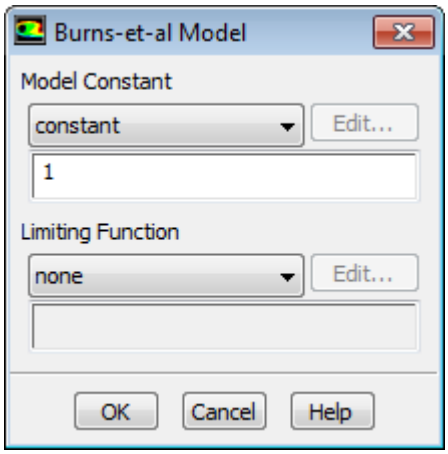
Specify the coefficient, C_{TD}, in Equation 17.280 in the Theory Guide. You can enter a constant value or specify a user-defined function defined using the DEFINE_EXCHANGE_PROPERTY macro (DEFINE_EXCHANGE_PROPERTY).

Limiting Function

Specify the limiting function to use. You may select **none**, the **standard** limiting function, or a **user-defined** function using the DEFINE_EXCHANGE_PROPERTY macro (DEFINE_EXCHANGE_PROPERTY). See Limiting Functions for the Turbulent Dispersion Force in Fluent Theory Guide for details on the limiting functions.

- Select **burns-et-al** to use the Burns et al. model (Burns et al. Model in the Theory Guide). You can edit the model parameters in the **Burns-et-al Model** dialog box (Figure 24.46: Burns et al. Model Dialog Box (p. 1472)).

Figure 24.46: Burns et al. Model Dialog Box



The following model parameters are available:

Model Constant

Specify the coefficient, C_{TD}, in Equation 17.282 in the Theory Guide. You can enter a constant value or specify a user-defined function defined using the DEFINE_EXCHANGE_PROPERTY macro (DEFINE_EXCHANGE_PROPERTY).

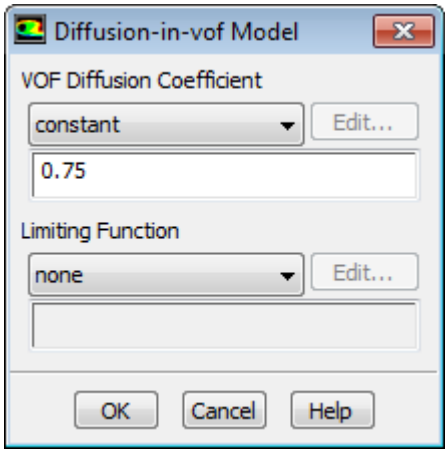
Limiting Function

Specify the limiting function to use. You may select **none**, the **standard** limiting function, or a **user-defined** function using the DEFINE_EXCHANGE_PROPERTY macro (DEFINE_EXCHANGE_PROPERTY). See Limiting Functions for the Turbulent Dispersion Force in Fluent Theory Guide for details on the limiting functions.

- Select **diffusion-in-vof** to use the Diffusion in VOF model (Diffusion in VOF Model in the Theory Guide). You can edit the model parameters in the **Diffusion-in-vof Model** dialog box (Figure 24.47: Diffusion—in—vof Model Dialog Box (p. 1473)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.47: Diffusion—in—vof Model Dialog Box



The following model parameters are available:

VOF Diffusion Coefficient
Specify the coefficient, σ_q, in Equation 17.285 in the Theory Guide.

Limiting Function
Specify the limiting function to use. You may select **none**, the **standard** limiting function, or a **user-defined** function using the DEFINE_EXCHANGE_PROPERTY macro (DEFINE_EXCHANGE_PROPERTY). See Limiting Functions for the Turbulent Dispersion Force in Fluent Theory Guide for details on the limiting functions.

- Select **user-defined** to use a user-defined function for the turbulent dispersion (see DEFINE_VECTOR_EXCHANGE_PROPERTY in the *Fluent Customization Manual* for details).

24.5.2.5.6. Including Surface Tension and Wall Adhesion Effects

As discussed in When Surface Tension Effects Are Important in the Theory Guide, the importance of surface tension effects depends on the value of the capillary number, Ca (defined by Equation 17.31 in the Theory Guide), or the Weber number, We (defined by Equation 17.32 in the Theory Guide). Surface tension effects can be neglected if $\mathrm{Ca} \gg 1$ or $\mathrm{We} \gg 1$.

Important

Note that the calculation of surface tension effects will be more accurate if you use a quadrilateral or hexahedral mesh in the area(s) of the computational domain where surface tension is significant. If you cannot use a quadrilateral or hexahedral mesh for the entire domain, then you should use a hybrid mesh, with quadrilaterals or hexahedra in the affected areas. ANSYS Fluent also offers an option to use VOF gradients at the nodes for curvature calculations on meshes when more accuracy is desired. For more information, see Surface Tension and Adhesion in the Theory Guide.

If you want to include the effects of surface tension along the interface between one or more pairs of phases, as described in Surface Tension and Adhesion in the Theory Guide, refer to Including Surface Tension and Adhesion Effects (p. 1438).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.5.2.5.7. Including the Virtual Mass Force

For both granular and non-granular flows, it is possible to include the "virtual mass force" ($\vec{F}_{vm}$ in Equation 17.288 in the Theory Guide) that is present when a secondary phase accelerates relative to the primary phase. The virtual mass effect is significant when the secondary phase density is much smaller than the primary phase density (for example, for a transient bubble column).

To include the effect of the virtual mass force perform the following steps:

1. Click the **Virtual Mass** tab to display the **Virtual Mass** inputs.

2. Enable **Virtual Mass Modeling**.

3. For each phase pair listed under **Virtual Mass Coefficient**, select the method for specifying the virtual mass coefficient, C_{vm}, in Equation 17.288 in the *Fluent Theory Guide*. You can choose from the following methods:

   none
   Disable virtual mass modeling for the phase pair.

   constant
   Specify a constant value for the coefficient. The default value of `0.5` is typical.

   user-defined
   Use a user-defined function created with the `DEFINE_EXCHANGE_PROPERTY` macro. See `DEFINE_EXCHANGE_PROPERTY` in the *Fluent Customization Manual* for details.

   Important

   No virtual mass effects will be included unless the **Virtual Mass Modeling** option is enabled.

4. If desired, you can enable the **Implicit** method for virtual mass.

   When using the **Implicit** method, the **Default** option uses the complete term in Equation 17.288. You can also choose **Option 2** or **Option 3**, which use locally truncated forms of the virtual mass terms in cells where divergence is detected.

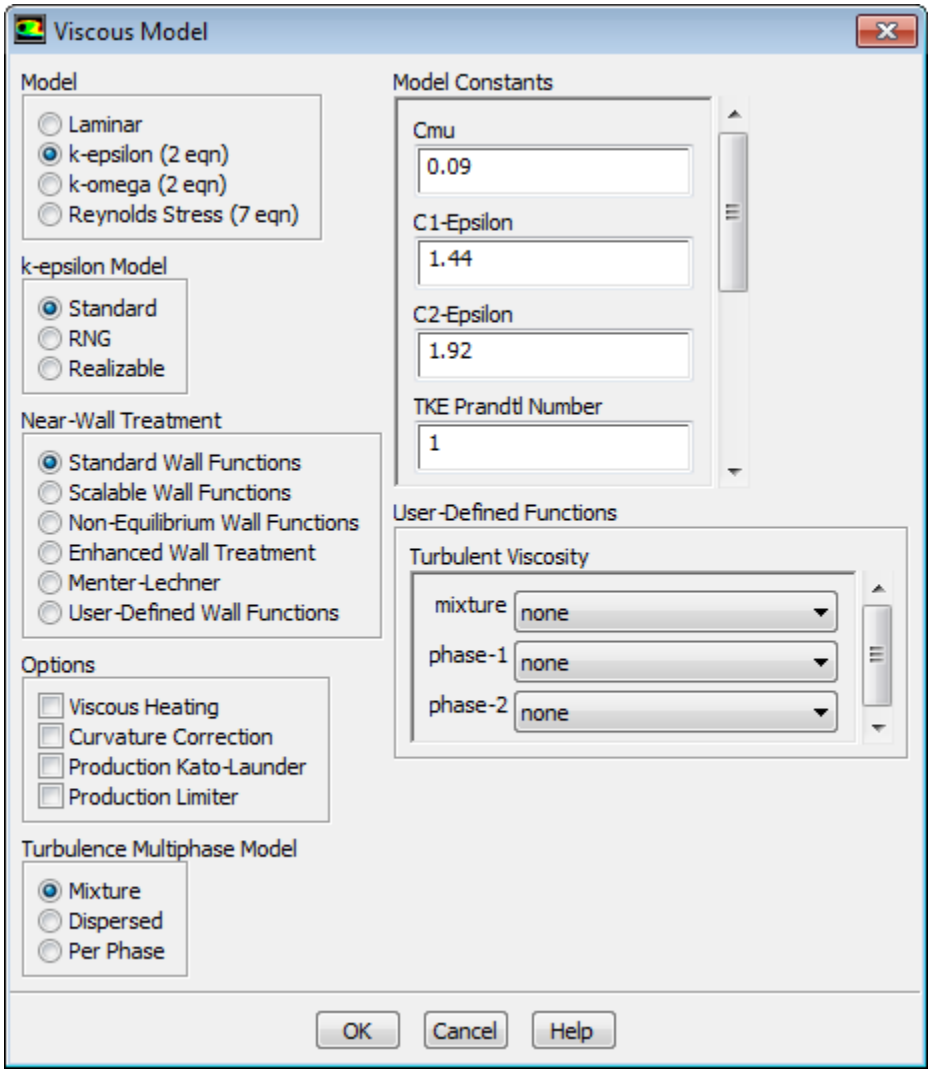
   The **Implicit** method is recommended for steady-state coupled simulations as it offers improved convergence in such cases. The recommended approach is to begin the simulation using **Option 2**. Once the solution has converged enough to establish the basic flow field, switch back to **Default** to include the complete virtual mass term.

24.5.3. Modeling Turbulence

If you are using the Eulerian model to solve a turbulent flow, you will need to choose one of turbulence models described in Turbulence Models in the Theory Guide in the Viscous Model Dialog Box (p. 2176) (Figure 24.48: The Viscous Model Dialog Box for an Eulerian Multiphase Calculation (p. 1475)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.48: The Viscous Model Dialog Box for an Eulerian Multiphase Calculation



The procedure is as follows:

1. Select **k-epsilon**, **k-omega**, or **Reynolds Stress** under **Model**.

2. Select the desired **k-epsilon Model**, **k-omega Model**, or **Reynolds-Stress Model** and any other related parameters, as described for single-phase calculations in Steps in Using a Turbulence Model (p. 764).

3. Under **Turbulence Multiphase Model** or **RSM Multiphase Model**, indicate the desired multiphase turbulence model (see Turbulence Models in the Theory Guide for details about each):

   - Select **Mixture** to use the mixture turbulence model. This is the default model.

   - Select **Dispersed** to use the dispersed turbulence model. This model is applicable when there is clearly one primary continuous phase and the rest are dispersed dilute secondary phases.

   - Select **Per Phase** to use a k-ε or k-ω turbulence model for each phase. This model is appropriate when the turbulence transfer among the phases plays a dominant role.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

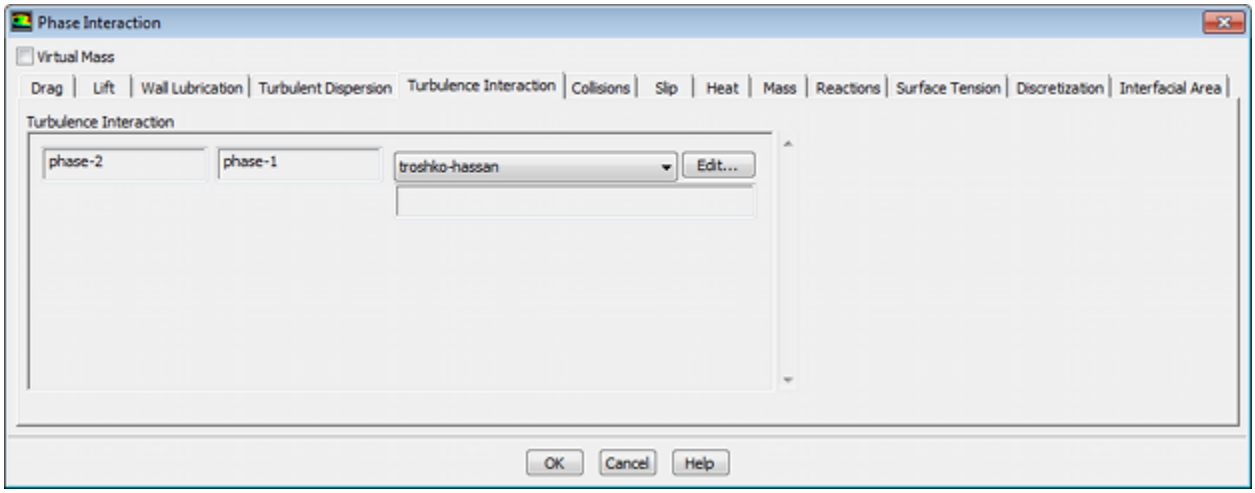
24.5.3.1. Including Turbulence Interaction Source Terms

By default, interphase turbulence source terms are not included in the calculation. If you want to include these source terms, you can enable them from the **Turbulence Interaction** tab of the Phase Interaction Dialog Box (p. 2363).

Setup → Models → Multiphase → Phases → Edit... → Interaction...

1. Click the **Interaction...** button to open the **Phase Interaction** dialog box (for example, Figure 24.49: The Phase Interaction Dialog Box for Turbulence Interaction (p. 1476)).

   Figure 24.49: The Phase Interaction Dialog Box for Turbulence Interaction



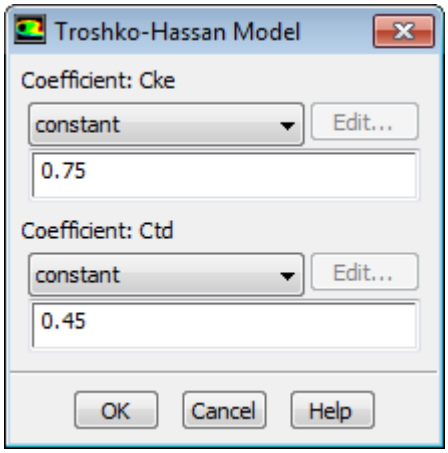
2. Click the **Turbulence Interaction** tab in the **Phase Interaction** dialog box.

3. For each pair of phases, select the desired correlation for the Turbulence Interaction. The available options are:

   - select **none** to omit turbulence interaction source terms. This is the default.
   - select **troshko-hassan** to use the Troshko-Hassan model described in Troshko-Hassan in the *Fluent Theory Guide*. You can edit the model parameters in the **Troshko-Hassan Model** dialog box (Figure 24.50: Troshko-Hassan Model Dialog Box (p. 1477)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.50: Troshko-Hassan Model Dialog Box



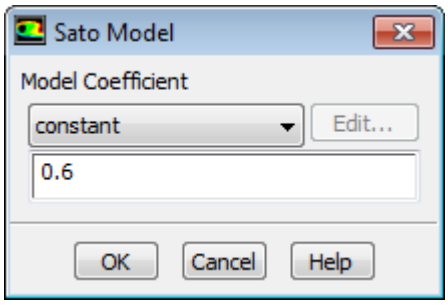
The following model parameters are available:

Coefficient: Cke
Specify the coefficient, C_{ke}, in the equations in Troshko-Hassan in the *Fluent Theory Guide*.

Coefficient: Ctd
Specify the coefficient, C_{td}, in the equations in Troshko-Hassan in the *Fluent Theory Guide*.

- select **sato** to use the Sato model described in Sato in the *Fluent Theory Guide*. You can edit the model parameters in the **Sato Model** dialog box (Figure 24.51: Sato Model Dialog Box (p. 1477)).

Figure 24.51: Sato Model Dialog Box



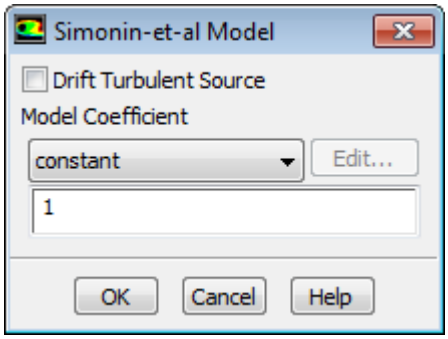
The following model parameters are available:

Model Coefficient
Specify the coefficient, $C_{\mu,p}$, in the equations in Sato in the *Fluent Theory Guide*.

- select **simonin-et-al** to use the Simonin et al. model described in Simonin et al. in the *Fluent Theory Guide*. This option is available only when **Dispersed** or **Per Phase** is selected in the **Viscous Model** dialog box. You can edit the model parameters in the **Simonin-et-al Model** dialog box (Figure 24.52: Simonin-et-al Model Dialog Box (p. 1478)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.52: Simonin-et-al Model Dialog Box



The following model parameters are available:

Drift Turbulent Source
include the portion the turbulent kinetic energy source term arising from the drift velocity.

Model Coefficient
Specify the coefficient, C_s, in the equations in Simonin et al. in the *Fluent Theory Guide*.

If you decide to enable turbulence interaction for multiple phase pairs, it is recommended that you avoid mixing the different models and that you select the same model for each phase pair with turbulence interaction enabled.

Note that the inclusion of these terms can slow down convergence noticeably. If you are looking for additional accuracy, you may want to compute a solution first without these sources, and then continue the calculation with these terms included. In most cases these terms can be neglected.

24.5.3.2. Customizing the k- ε Multiphase Turbulent Viscosity

If you are using the k- ε multiphase turbulence model, a user-defined function can be used to customize the turbulent viscosity for each phase. This option will enable you to modify μ_t in the k- ε model. For more information, see the Fluent Customization Manual.

In the **Viscous Model** dialog box, under **User-Defined Functions**, select the appropriate user-defined function in the **Turbulent Viscosity** drop-down list.

24.5.4. Including Heat Transfer Effects

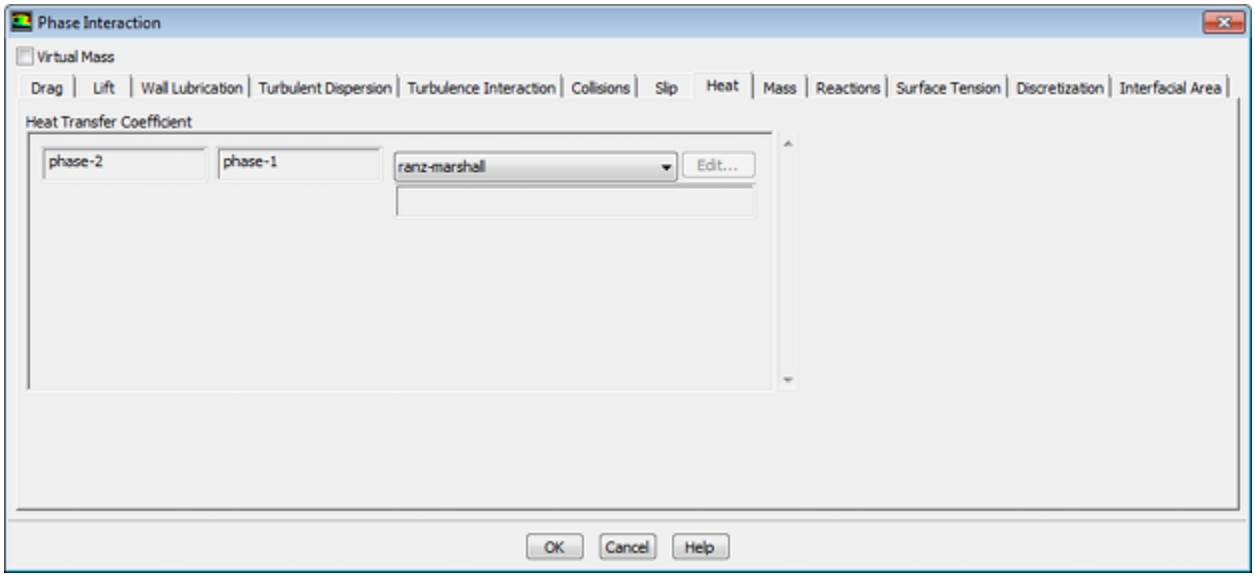
To define heat transfer in a multiphase Eulerian simulation, you will need to visit the **Phase Interaction** dialog box, after you have enabled the energy equation in the **Energy** dialog box.

Setup → Models → Multiphase → Phases → Edit... → Interaction...

1. Click the **Interaction...** button to open the **Phase Interaction** dialog box (for example, Figure 24.53: The Phase Interaction Dialog Box for Heat Transfer (p. 1479)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.53: The Phase Interaction Dialog Box for Heat Transfer



2. Click the **Heat** tab in the **Phase Interaction** dialog box.

3. Select the desired correlation for the **Heat Transfer Coefficient**. The available choices will depend upon what other models you have enabled. Note the following regarding the available choices:

   constant-htc
   allows you to specify a constant value for the volumetric heat transfer coefficient.

   nusselt-number
   allows you to specify a value for the Nusselt number from which the heat transfer coefficient will be computed.

   gunn
   is frequently used for Eulerian multiphase simulations involving a granular phase.

   ranz-marshall
   is frequently used for Eulerian multiphase simulations not involving a granular phase.

   hughmark
   is an extension of Ranz-Marshall to a wider range of Reynolds Number.

   tomiyama
   is frequently used for Eulerian multiphase simulations of bubbly flows with relatively low Reynolds number.

   none
   allows you to ignore the effects of heat transfer between the two phases.

   user-defined
   allows you to implement a correlation reflecting a model of your choice, through a user-defined function.

   two-resistance
   allows you to independently specify the heat transfer coefficient correlations for the two phases. This setting is recommended when using the evaporation-condensation mass transfer model.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

fixed-to-sat-temp
models heat transfer when all of the heat transfer to a phase interface is used in mass transfer. It assumes that the temperature at the **To-Phase** in the mass transfer is equal to the saturation temperature. Note that this model only applies when one of the mass transfer models is enabled.

lavieville-et-al
is only available within the two-resistance formulation and can be used to model the gas-interface heat transfer coefficient. It assumes that the gas phase retains the saturation temperature by rapid evaporation or condensation.

zero-resistance
is only available within the two-resistance formulation. It may be specified for one phase in which case that phase temperature is equal to the interfacial temperature.

4. Set the appropriate thermal boundary conditions. You will specify the thermal boundary conditions for each individual phase on most boundaries, and for the mixture on some boundaries. See Cell Zone and Boundary Conditions (p. 237) for more information on boundary conditions, and Eulerian Model (p. 1401) for more information on specifying boundary conditions for a Eulerian multiphase calculation.

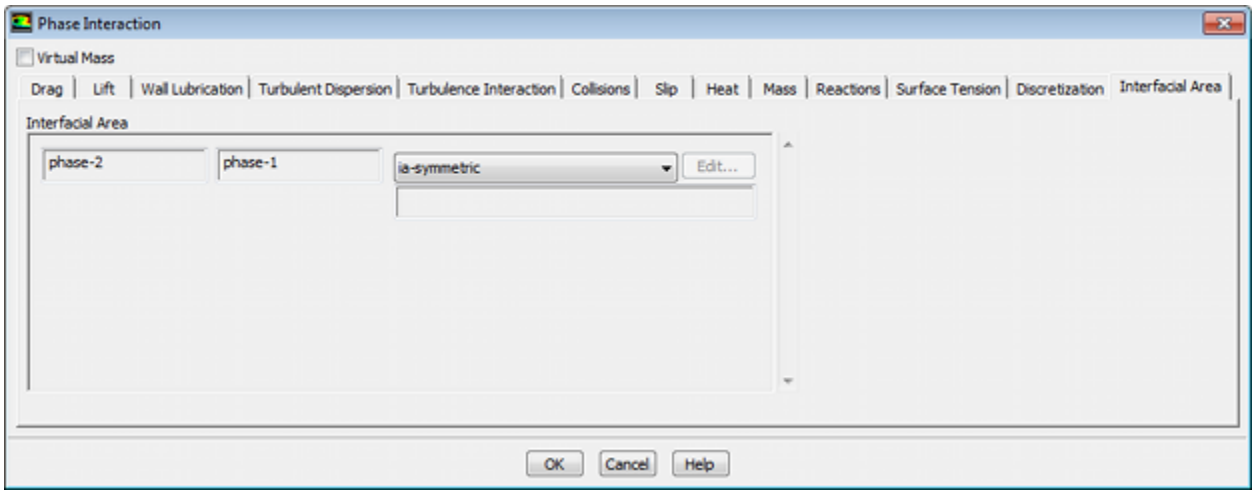
See Description of Heat Transfer in the Theory Guide for more information on heat transfer in the framework of a Eulerian multiphase simulation and the available models.

24.5.5. Using an Algebraic Interfacial Area Model

If you have chosen not to solve the transport equation for Interfacial Area Concentration (Defining the Interfacial Area Concentration (p. 1461)), you can select an algebraic model to estimate the interfacial area from the secondary phase diameter specified in the Secondary Phase Dialog Box (p. 2357). To choose an algebraic interfacial area model perform these steps.

1. Click the **Interaction...** button to open the **Phase Interaction** dialog box (for example, Figure 24.54: The Phase Interaction Dialog Box for Interfacial Area (p. 1480)).

Figure 24.54: The Phase Interaction Dialog Box for Interfacial Area



2. Click the **Interfacial Area** tab in the **Phase Interaction** dialog box.

3. Select the desired algebraic model for the **Interfacial Area**. Note the following regarding the available choices:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ia-symmetric
considers both the primary and secondary phase volume fractions in estimating the interfacial area.

ia-particle
considers only the secondary phase volume fraction in estimating the interfacial area.

Additional options are available if you have enabled one of the boiling models. See Including the Boiling Model (p. 1486).

See Interfacial Area Concentration in the *Fluent Theory Guide* for details about the algebraic interfacial area models.

24.5.6. Including the Dense Discrete Phase Model

If you are using the Eulerian multiphase model (Setting Up the Eulerian Model (p. 1455)), you have the option of including the **Dense Discrete Phase Model** (Dense Discrete Phase Model in the Theory Guide).

Important

- Enabling this model automatically enables the DPM model. You will notice that **Interaction with Continuous Phase** in the **Discrete Phase Model** dialog box is enabled.
- The Dense Discrete Phase model is available only with the **Eulerian** multiphase model. The limitations that currently apply to DPM and Eulerian multiphase models also apply to the Dense Discrete Phase model. See Limitations of the Eulerian Model in the *Fluent Theory Guide* and Limitations (p. 1238) for limitations that exist with the DPM and Eulerian multiphase models.

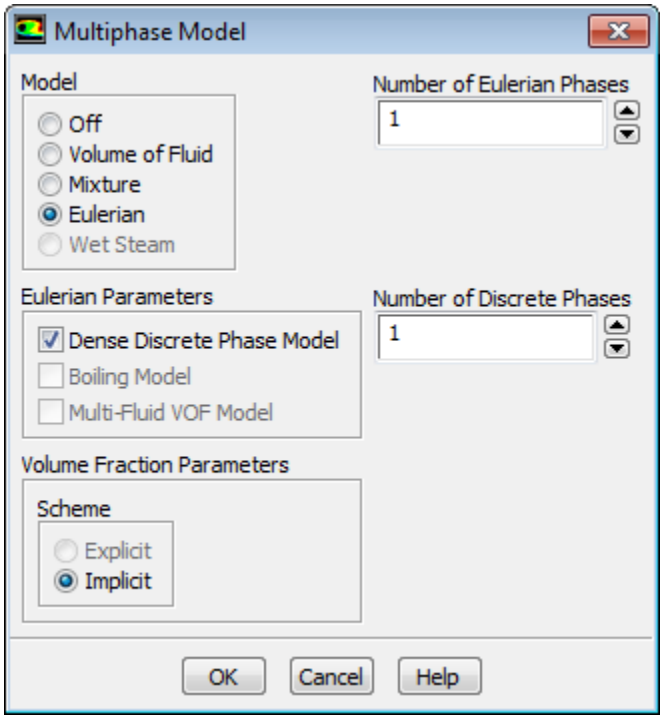
The required work flow when using the dense discrete phase model is as follows:

1. Set up the **Multiphase Model** dialog box (Figure 24.55: The Dense Discrete Phase Model (p. 1482)) to include the dense discrete phase model parameters.

   Setup → **Models** → **Multiphase** → **Edit...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.55: The Dense Discrete Phase Model



a. Enable **Dense Discrete Phase Model** under **Eulerian Parameters**.

b. Set the **Number of Discrete Phases** that are present in your case.

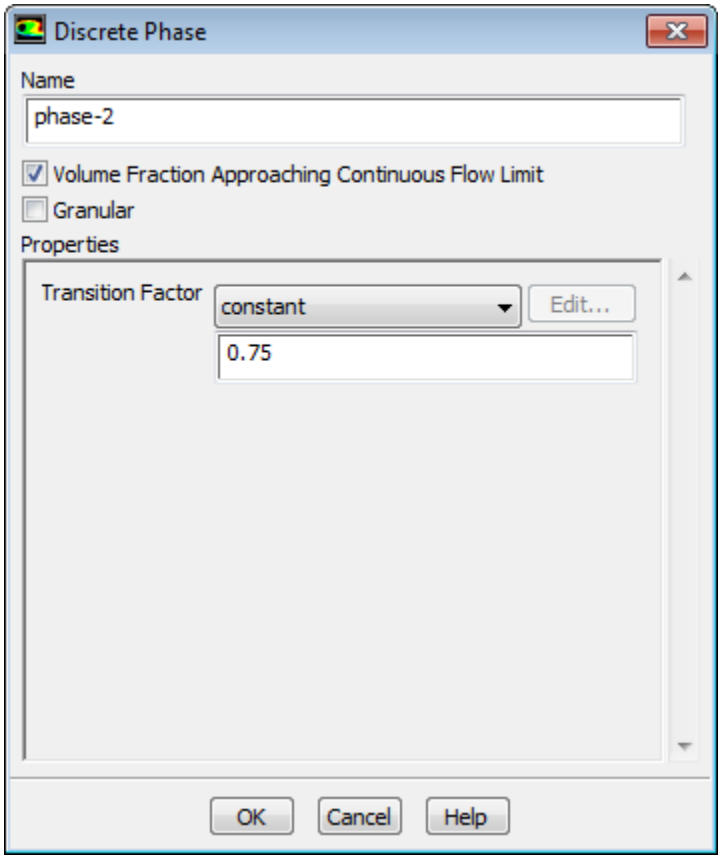
2. Open the **Phases** dialog box, in order to define the phases.

Setup → Models → Multiphase → Phases → Edit...

a. Define the discrete phase, by selecting the phase from the **Phases** selection list that is labeled **Discrete Phase** and clicking the **Edit...** button. Then set up the properties in the **Discrete Phase** dialog box that opens, as shown in Figure 24.56: The Discrete Phase Dialog Box (p. 1483).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.56: The Discrete Phase Dialog Box



Note that for non-granular flow, no additional inputs are required here (since the material is set automatically in the background, and the diameter is part of the solution).

b. Enable **Volume Fraction Approaching Continuous Flow Limit** to specify a **Transition Factor**. The **Transition Factor** is multiplied by the theoretical close-packing limit for mono-sized spheres ($\pi/3\sqrt{2} \approx 3/4$) to determine the transition volume fraction. The default value for the **Transition Factor** is 0.75, giving a transition volume fraction of 0.5625 ($=0.75*3/4$). Note that the **Transition Factor** is not limited to a range between 0 and 1. In other words, you can specify values outside this range. For example, a value of 1.2 will give a transition volume fraction of 0.9. You can also specify locally variable values using the `DEFINE_PROPERTY` user-defined macro, depending on the local particle size distribution, as an example.

See `DEFINE_PROPERTY` UDFs in the Fluent Customization Manual for information about the `DEFINE_PROPERTY`.

c. Define the primary and secondary phases, as described in Defining the Phases for the Eulerian Model (p. 1456).

3. Define the injections using the **Injections** dialog box.

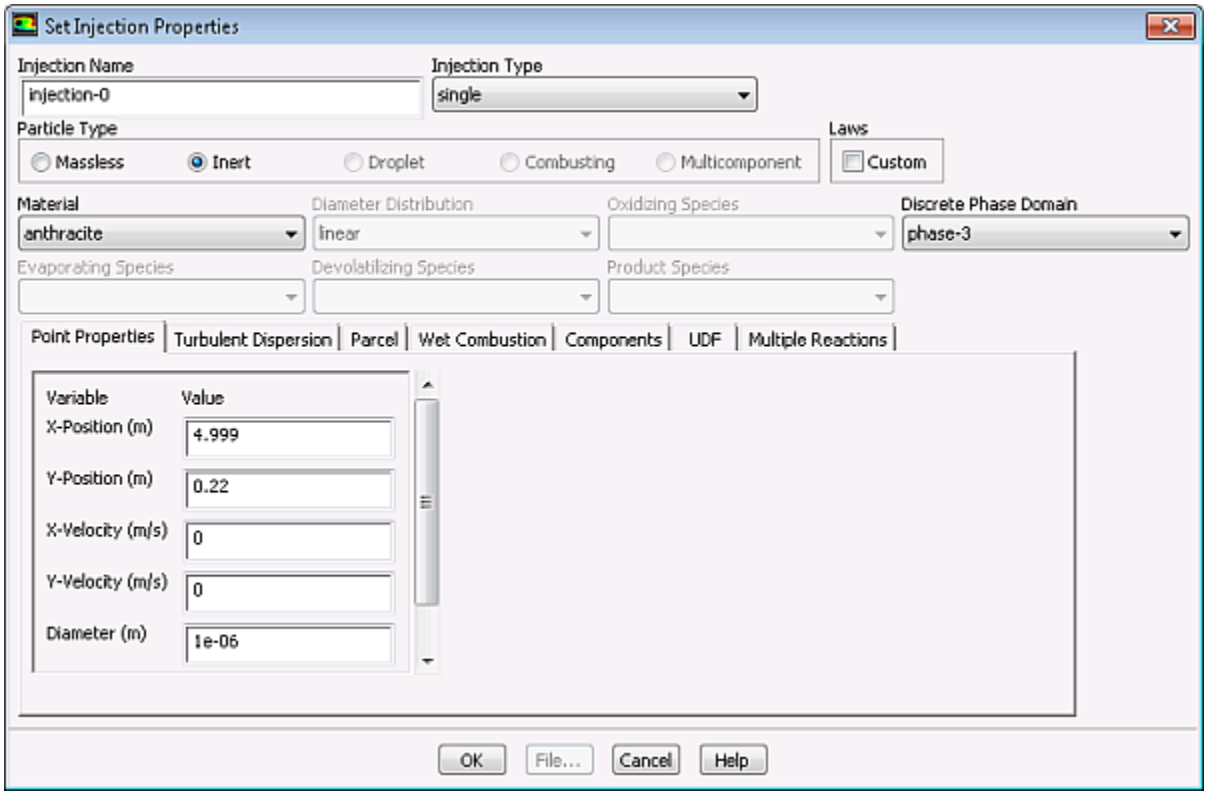
Setup → Models → Discrete Phase → Injections → New...

a. Create a new injection by clicking the **Create** button in the **Injections** dialog box, or edit an existing injection by selecting the injection from the **Injections** list and clicking the **Set...** button.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. In the **Set Injection Properties** dialog box that opens (Figure 24.57: The Set Injection Properties Dialog Box (p. 1484)), specify properties for each injection as described in Setting Initial Conditions for the Discrete Phase (p. 1265)

Figure 24.57: The Set Injection Properties Dialog Box



Note

Note that when the DDPM is used, **averaged-discrete-phase-drag** is automatically selected for the drag coefficient in the **Drag** tab of the **Phase Interaction** dialog box. **averaged-discrete-phase-drag** corresponds to K_{DPM} in Equation 17.403 in the *Fluent Theory Guide* and indicates that the interaction between discrete and fluid phases will be computed based on averaged values of fluid drag of all particles crossing the fluid cell. That is, at each time step, the drag is evaluated for each particle using the drag law that you have selected in the **Set injection Properties** dialog box and then averaged based on the residence time of each particle in the cell.

c. Select the discrete phase from the **Discrete Phase Domain** drop-down list.

4. Define the material properties for each injection.

Setup → Materials

24.5.6.1. Defining a Granular Discrete Phase

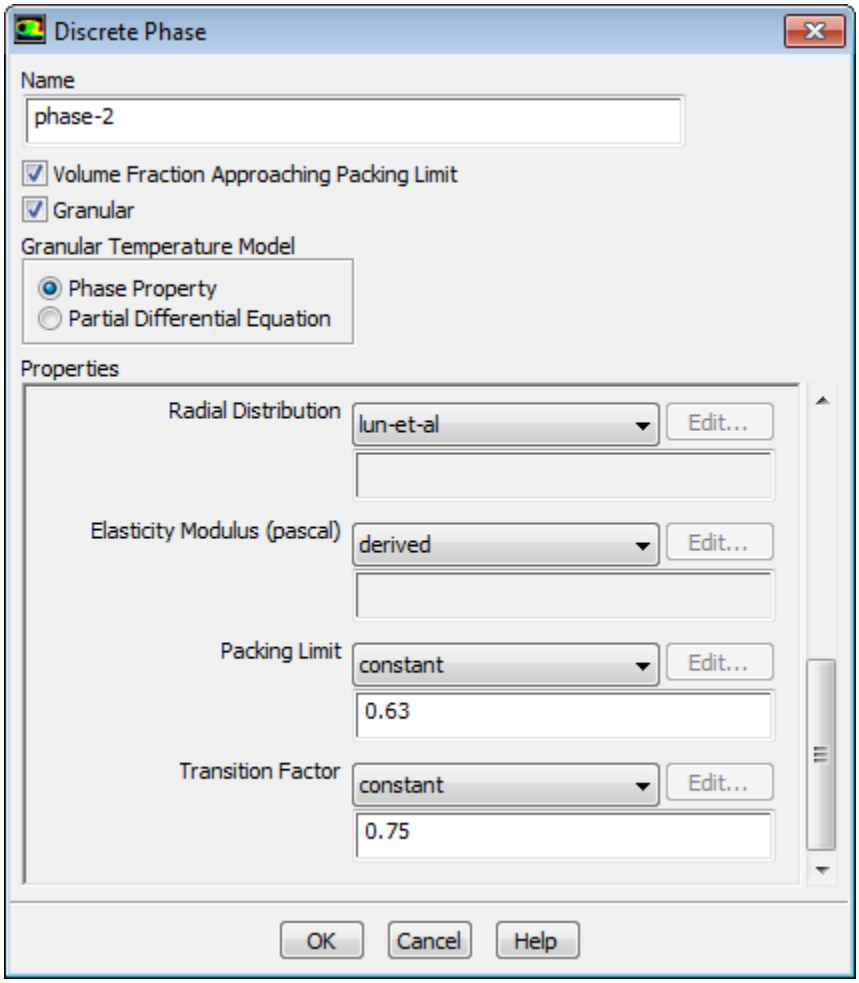
To define a granular (that is, particulate) discrete phase in an Eulerian multiphase calculation, perform the following steps:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the phase (for example, **Discrete Phase**) in the **Phases** list.

2. Click **Edit...** to open the **Discrete Phase** dialog box (Figure 24.58: The Discrete Phase Dialog Box for a Granular Phase (p. 1485)).

Figure 24.58: The Discrete Phase Dialog Box for a Granular Phase



3. In the **Discrete Phase** dialog box, enter a **Name** for the phase.

4. Enable the **Granular** option.

5. Enable **Volume Fraction Approaching Packing Limit** to prevent the unlimited accumulation of particles, which are operating at packing limit conditions.

6. Specify the **Transition Factor** as either a **constant** or a **user-defined** function. The default value for the **Transition Factor** is 0.75. The transition criterion is based on the local particle volume fraction of the given discrete phase and is specified as a factor multiplied by the maximum packing limit (also a user-specified value). For example, for a typical granular phase with a maximum packing limit of 0.63, the transition volume fraction is the product of 0.75 and 0.63, which is equal to 0.4725.

You can now define all other fields of the discrete phase in a similar manner to that described in Defining a Granular Secondary Phase (p. 1457).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.5.7. Including the Boiling Model

If you are using the Eulerian multiphase model (Setting Up the Eulerian Model (p. 1455)), you have the option of including the **Boiling Model** (see Wall Boiling Models in the Theory Guide).

Note the following limitations:

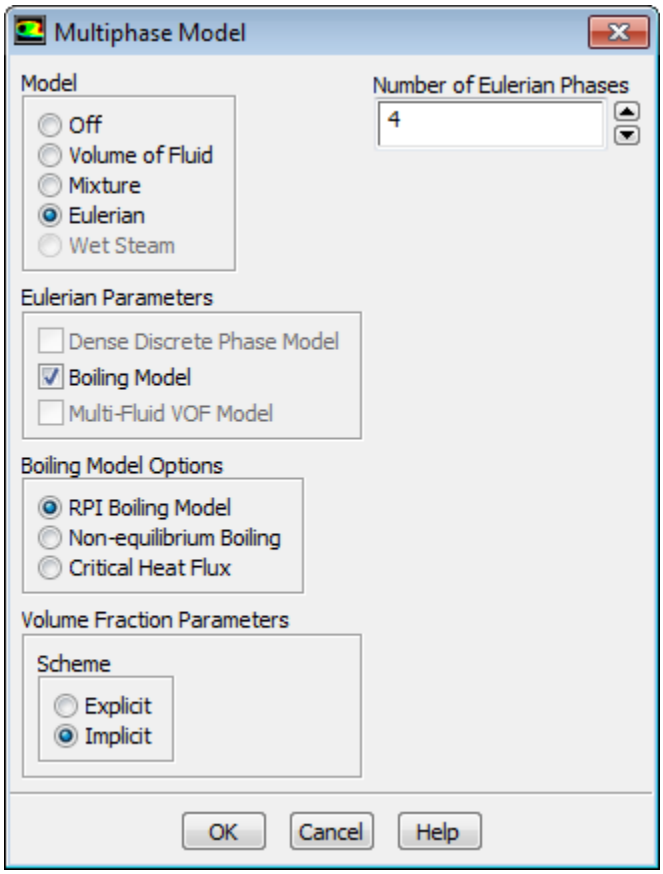
- This model is only available with the **Eulerian** multiphase model.
- This model is only available with the **Pressure-Based** solver.

The required work flow when using the boiling model is as follows:

1. Open the **Multiphase Model** dialog box (Figure 24.59: The Boiling Model (p. 1486)).

   Setup → **Models** → **Multiphase** → **Edit...**

   Figure 24.59: The Boiling Model



   a. Enable **Boiling Model** under **Eulerian Parameters**.

   b. Select **RPI Boiling Model**, **Non-equilibrium Boiling**, or **Critical Heat Flux** as the boiling model option. Information about the three options is available in RPI Model, Non-equilibrium Subcooled Boiling, and Critical Heat Flux in the Theory Guide.

   c. Set the total **Number of Eulerian Phases** that are present in your case. This can consist of two phases: liquid and vapor, which are directly involved in boiling mass transfer; or it can include "non-boiling

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

phases" or "species". If a system consists of three phases: liquid, vapor, and air, then the **Number of Eulerian Phases** will be 3, where air is the non-boiling phase.

Note

- To include the liquid volume fraction effects, use the `solve/set/multiphase-numeric/boiling-parameters/liquid-vof-factor` text command. When this option is enabled, the heat transfer coefficients of boiling models will be multiplied by the local liquid volume fraction.
- To include the thin film effects, use the `solve/set/multiphase-numeric/boiling-parameters/thin-film` text command. When enabled, this option will use Equation 17.433 in the *Fluent Theory Guide* in the calculation.

2. Make sure the energy option is enabled.

Note

The energy option will be automatically turned on when the boiling model is enabled.

Setup → Models → Energy → On

3. Enable **Gravity** and set the **Operating Pressure** in the **Operating Conditions** dialog box.

Important

Make sure gravity is included when the boiling model is used.

4. Choose one of the turbulence models that is available with the Eulerian multiphase model.

Setup → Models → Viscous → Edit...

5. Open the **Create/Edit Materials** dialog box and specify the material properties for the liquid, vapor, solid, and any other phases that may exist.

Setup → Materials → Fluid → Create/Edit...

Important

For the liquid and vapor phases, the **Standard State Enthalpy** must be specified, as it is used in the computation of the **Latent Heat**.

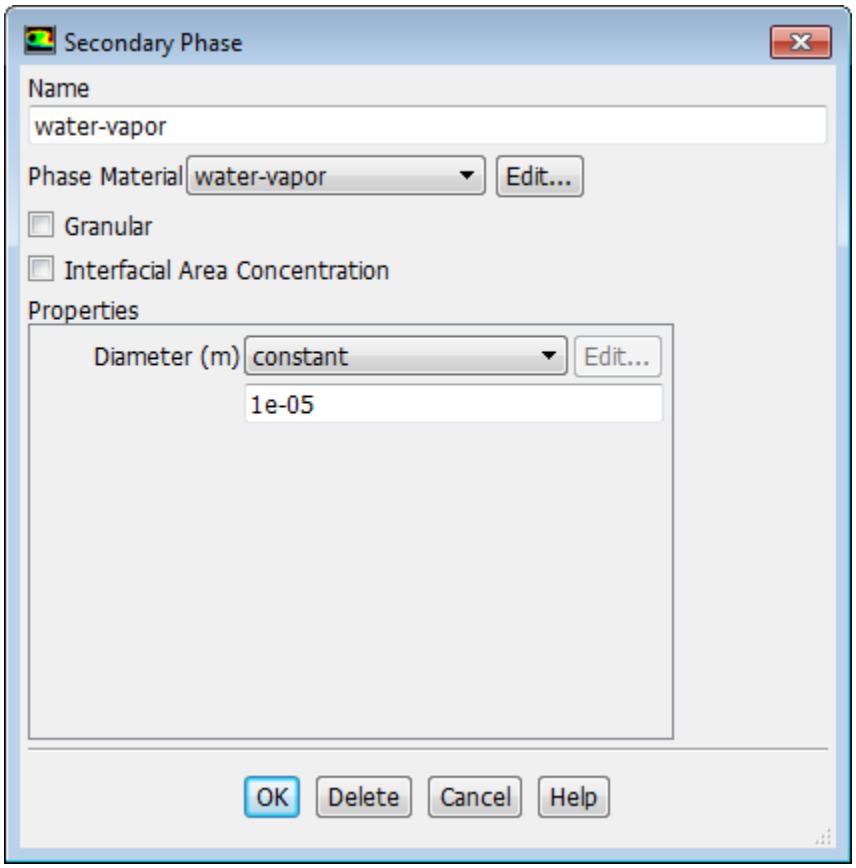
6. Open the **Phases** dialog box, in order to define the phases.

Setup → Models → Multiphase → Phases → Edit...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Define the liquid as the primary phase by selecting the phase from the **Phases** selection list that is labeled **Primary Phase** and clicking the **Edit...** button. Define the vapor as the secondary phase. In the **Secondary Phase** dialog box, the **Diameter** is set to **boiling-dia** by default, but you have the option of setting it up as a **constant** value or a **user-defined** function, as shown in Figure 24.60: The Secondary Phase Dialog Box (p. 1488).

Figure 24.60: The Secondary Phase Dialog Box



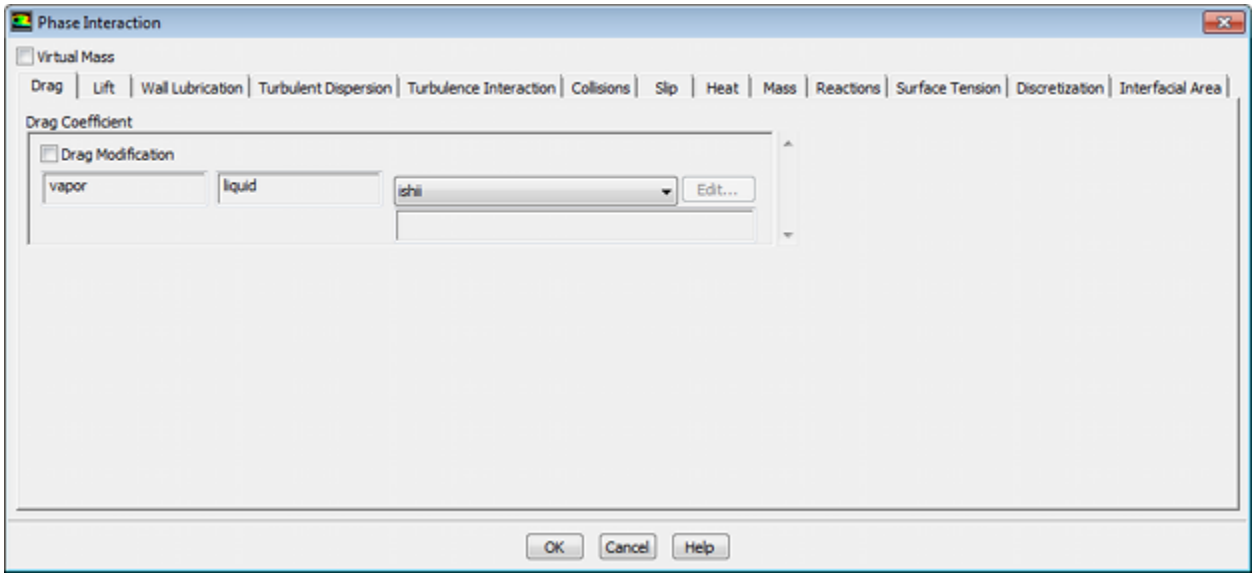
7. Open the **Phase Interaction** dialog box.

   Setup → **Models** → **Multiphase** → **Phases** → **Edit...** → **Interaction...**

   a. In the **Drag** tab, you have a choice of **ishii**, **universal-drag**, **schiller-naumann**, **symmetric**, **morsi-alexander**, **grace**, **tomiyama**, or **user-defined**. See Specifying the Drag Function (p. 1463) for definitions of the drag options For boiling flows, **ishii** is usually chosen.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.61: The Phase Interaction Dialog Box



b. In the **Lift** tab, select **moraga**, **tomiyama**, **saffman-mei**, **legendre-magnaudet**, or **user-defined**. For boiling flows, **tomiyama** is usually chosen.

c. In the **Wall Lubrication** tab, select **antal-et-al**, **tomiyama-et-al**, **frank**, **hosokawa**, or **user-defined** as described in Including the Wall Lubrication Force (p. 1467). For boiling flows, **antal-et-al** is typically chosen.

d. In the **Turbulent Dispersion** tab, select **lopez-de-bertodano**, **simonin**, **burns-et-al**, **diffusion-in-vof**, or **user-defined** as described in Including the Turbulent Dispersion Force (p. 1470). For boiling flows, **lopez-de-bertodano** is typically chosen.

e. In the **Turbulent Interaction** tab, select **troshko-hassan**, **sato**, or **simonin-et-al** as described in Including Turbulence Interaction Source Terms (p. 1476). For boiling flows, **troshko-hassan** is typically chosen.

f. In the **Heat** tab, select **ranz-marshall**, **tomiyama**, or **user-defined**. For boiling flows, **ranz-marshall** is usually chosen.

g. In the **Surface Tension** tab, select **constant** and enter the desired value.

h. In the **Interfacial Area** tab, you have four formulations available from which to select:

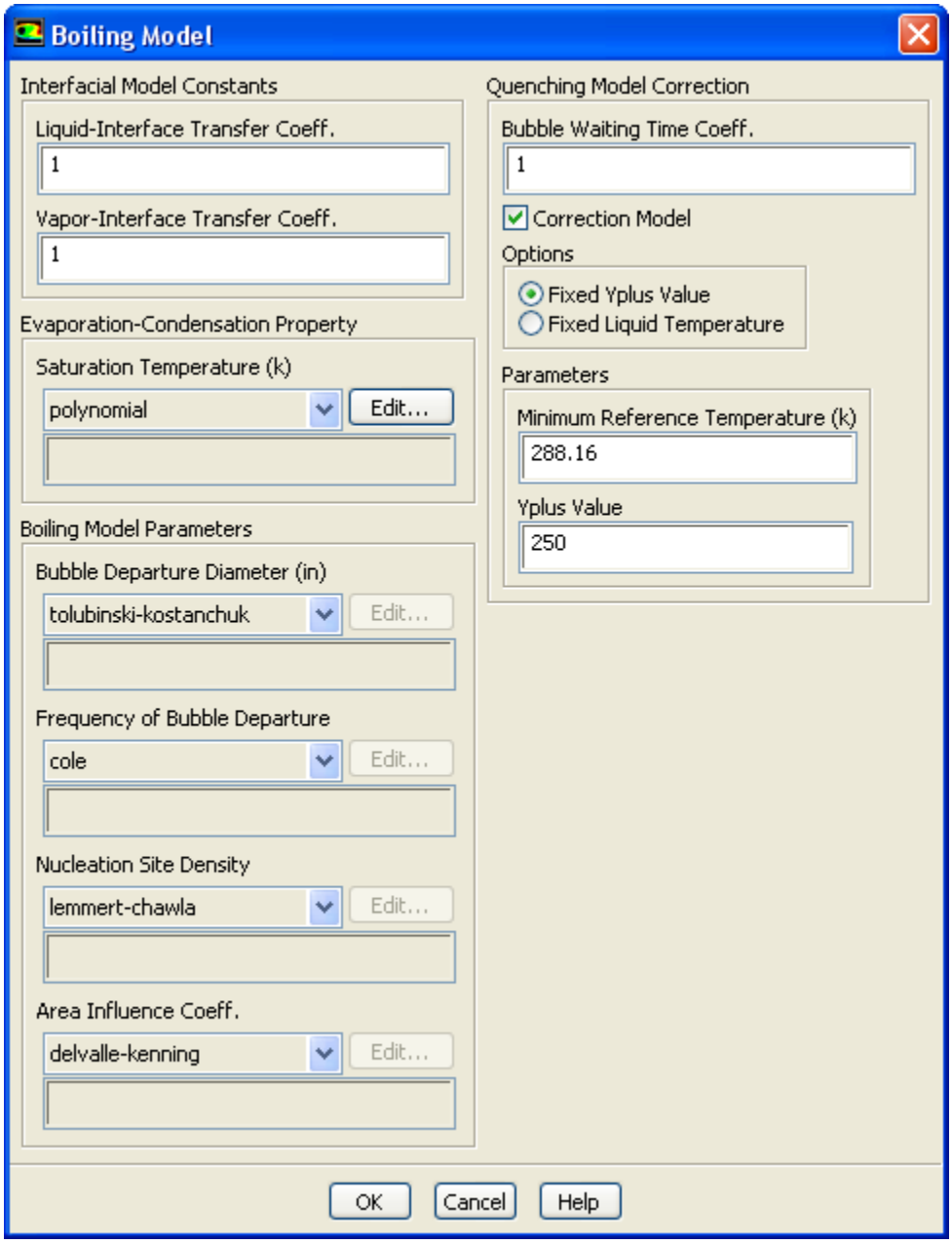
- **ia-symmetric** (default) (see Equation 17.177 in the Theory Guide)
- **ia-particle** (see Equation 17.176 in the Theory Guide)
- **ia-ishii** (see Equation 17.178 in the Theory Guide). This option is only available with the RPI model.
- **user-defined** (see Example 4- Custom Interfacial Area in the Fluent Customization Manual)

For boiling flows, **ia-symmetric** or **ia-particle** are usually chosen.

i. In the **Mass** tab, set the **Number of Mass Transfer Mechanisms** to 1 and make sure the transfer is always from the liquid to the vapor phase. Select **boiling** under **Mechanism**. The **Boiling Model** dialog box will open (Figure 24.62: The Boiling Model Dialog Box (p. 1490)), where you will set the boiling model parameters and the quenching model corrections.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.62: The Boiling Model Dialog Box



i. Specify the **Interfacial Model Constants** and the **Saturation Temperature**. The default values for the liquid and vapor interface transfer coefficients are 1.

Note

- If you use the **polynomial**, **piecewise-polynomial**, or **piecewise-linear** option for **Saturation Temperature**, the pressure-dependency must be specified in terms of absolute pressure.
- If you selected the **RPI Boiling Model** in the **Multiphase Model** dialog box, then you will not need to specify the **Vapor-Interface Transfer Coeff.**, since the vapor phase temperature is fixed to the saturation temperature.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ii. Under the **Boiling Model Parameters**, select the **Bubble Departure Diameter** that best describes your model. Five options exist: **tolubinski-kostanchuk** (the default setting), **unal**, **kocamustafaogullari-ishii**, **constant**, and **user-defined**. The **tolubinski-kostanchuk** formulation is described in Equation 17.421 in the Theory Guide, the **unal** formulation is described in Equation 17.423 in the Theory Guide and the **kocamustafaogullari-Ishii** formulation is described in Equation 17.419 in the Theory Guide.

iii. Specify the **Frequency of Bubble Departure**. You can choose the **cole** option (which is the default), or enter a **constant** value. The **cole** formulation is described in Equation 17.417 in the Theory Guide.

iv. Specify the **Nucleation Site Density**. You can choose between **lemmert-chawla** (which is the default), **kocamustafaogullari-ishii**. and **user-defined**. This quantity is usually represented by a correlation based on the wall superheat, described in Equation 17.418 in the Theory Guide.

Note

It is recommended that when using the **kocamustafaogullari-ishii** option, it should be used for both the **Bubble Departure Diameter** and **Nucleation Site Density**.

---

v. Specify the **Area Influence Coeff.** The area of influence is based on the bubble departure diameter and the nucleate site density, as defined in Equation 17.414 in the Theory Guide. You have a choice of three options when modeling the area of influence coefficient: **delvalle-kenning** (which is the default and defined in Equation 17.415 in the Theory Guide, **constant**, and **user-defined**.

vi. The **Quenching Correction Model** addresses the quenching term in the wall heat flux partition (described in Wall Heat Flux Partition in the Theory Guide). The quenching term in the wall heat flux partition models the cyclic averaged transient energy transfer related to liquid filling the wall vicinity after the bubble detachment with a period T and it is expressed as:

$$\dot{q}_q = C_{wt} \frac{2k_l}{\sqrt{\pi \gamma_l T}} \left(T_w - T_l\right) A_b \tag{24.15}$$

where k_l is the liquid heat conductivity, T is the periodic time, and $\gamma_l = k_l \Big/ \left(\rho_l c_{pl}\right)$ is the liquid phase diffusivity. The **Bubble Waiting Time Coefficient**, C_{wt}, is a coefficient introduced to correct the waiting time between departures of consecutive bubbles. The default value is 1, however you can modify this value as needed, but it can only be specified as a constant.

From Equation 24.15 (p. 1491), you can see that the quenching model is strongly dependent on T_l, which results in grid-dependent solutions. To remedy this, two approaches have been adopted in ANSYS Fluent: **Fixed Yplus Value** and **Fixed Liquid Temperature**.

vii. Enable the **Correction Model** if you want to achieve a certain level of gird-independence in your solution. If this option is disabled, Equation 24.15 (p. 1491) calculates the quench flux term without corrections.

viii. Select **Fixed Yplus Value** if you want to use the logarithmic form of the wall functions to estimate the liquid temperature T_l at a fixed Yplus value of 250, as proposed by Egorov and Mentor [27] (p. 2888), instead of using the liquid temperature values in the near-wall cells. Enter the **Minimum Reference Temperature**, which limits the lowest value of the liquid temperature. It should not be lower than the liquid inlet temperature. Specify a **Yplus Value**. It is set to 250 by default.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ix. If you select **Fixed Liquid Temperature**, you can choose from **standard**, **constant**, or **user-defined** for the **Liquid Reference Temperature**. You will also need to enter the **Minimum Reference Temperature**. The liquid temperature used to compute the quenching flux should usually be between the inlet liquid temperature and the saturation temperature.

Note

To learn how to hook boiling parameter user-defined functions, refer to `DEFINE_BOILING_PROPERTY` in the Fluent Customization Manual.

8. Set up the conditions at inlets, outlets, and thermal conditions for walls.

Setup → **Boundary Conditions**

For the quenching wall heat flux, Koncar et al. [54] (p. 2890) have suggested that in order to avoid grid dependence when calculating the quenching heat transfer, a factor that relates the temperature at a fixed normalized distance (y+ = 250) to the temperature at the near wall cell must be applied. Contact a technical support engineer for more information.

The wall boiling models are compatible with three different wall boundaries: isothermal wall, specified heat flux, and specified heat transfer coefficient (coupled wall boundary).

Note

The boiling models do not apply to thin walls.

9. Choose **Coupled** as the pressure-velocity coupling scheme.

Solution → **Methods**

Note

Although **Phase Coupled SIMPLE** is also available, it is generally less robust and is not recommended for use in steady cases involving boiling or mass transfer.

10. The following solution strategies are recommended for boiling model simulations:

Solution → **Controls**

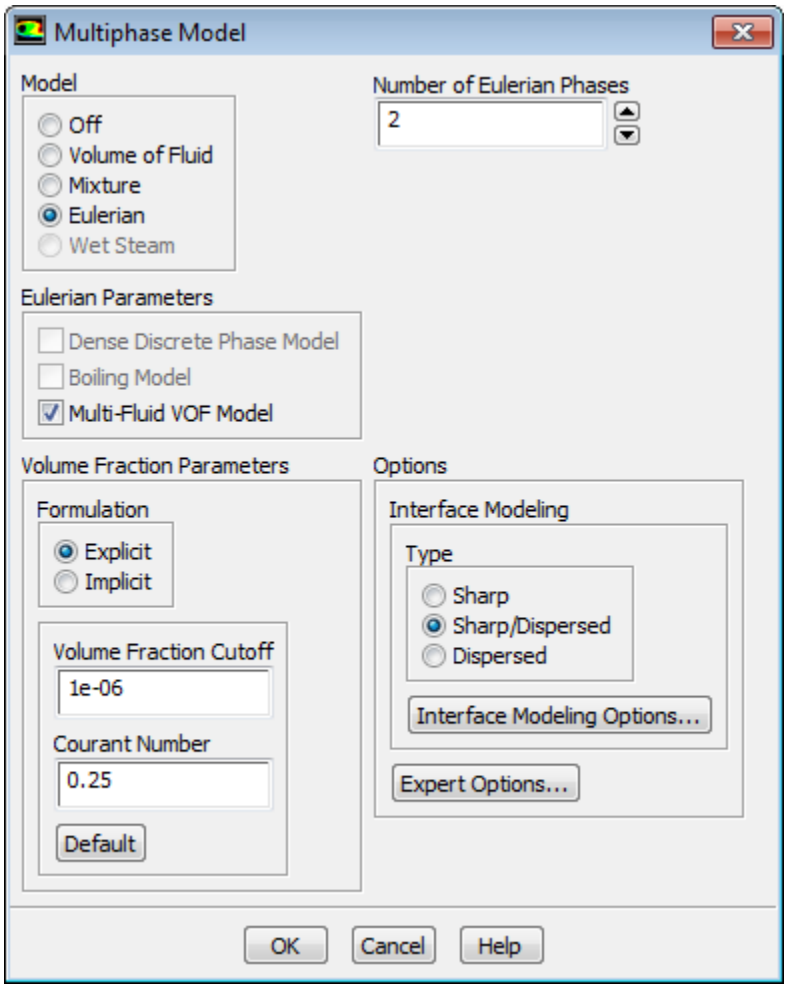
- **Courant Number**: use a value between 1 and 20 is recommended. As a first try, use a value of 10.
- **Explicit Relaxation Factors**: use the default values of 1 for **Momentum** and **Pressure**.
- **Vaporization Mass**: use an under-relaxation factor between 0.5 and 1. As a first try, use a value of 1.
- **Volume Fraction**: use an under-relaxation factor between 0.3 and 0.5.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Turbulent Kinetic Energy**: use an under-relaxation factor between 0.3 and 0.8.
- **Turbulent Viscosity**: use an under-relaxation factor between 0.5 and 1.0.
- **Energy**: use an under-relaxation factor between 0.5 and 0.8.

24.5.8. Including the Multi-Fluid VOF Model

After you have selected the Eulerian multiphase model (Setting Up the Eulerian Model (p. 1455)), you can enable the **Multi-Fluid VOF Model** (Multi-Fluid VOF Model in the Theory Guide).



Note the following limitations:

- This model is only available with the **Eulerian** multiphase model.
- You cannot use this model with the **Dense Discrete Phase Model**.

After enabling **Multi-Fluid VOF**, the **Interface Modeling Options** become available, as described in Interface Modeling Type (p. 1371). You can select from **Sharp**, **Dispersed**, or hybrid **Sharp/Dispersed** interface modeling to expose the most appropriate discretization schemes for your simulation. You can also specify some additional time-advancement settings as described in Setting Time-Dependent Parameters for the Explicit Volume Fraction Formulation (p. 1445).

The available drag laws depend on your selection for **Interface Modeling**. **Sharp** interface modeling supports only **symmetric**, **anisotropic-drag**, **user-defined**, and **none**. **Sharp/Dispersed** interface

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

modeling supports all the relevant drag laws. **Dispersed** interface modeling does not support **anisotropic-drag**, but supports all other relevant drag laws.

Note

Anisotropic drag applicable to free surface modeling is only compatible with the **Mixture** turbulence model.

If you want to use the anisotropic drag law, perform the following steps:

1. Define the drag law in the **Phase Interaction** dialog box.

   Setup → **Models** → **Multiphase** → **Phases** → **Edit...** → **Interaction...**

2. Click the **Drag** tab to display the **Drag Coefficient** inputs.

3. For each pair of phases, select the appropriate drag law from the corresponding drop-down list.

   - Select **anisotropic-drag** when there is higher drag in the normal direction to the interface and lower drag in the tangential direction to the interface. For details about this drag law, refer to Multi-Fluid VOF Model in the Theory Guide.

     To specify the input parameters for anisotropic drag, you will need to use the `/solve/set/mp-mfluid-aniso-drag` text command. The options for an `Anisotropic Drag Method` of `0` (which is based on the symmetric drag), are as follows:

     ```
     Anisotropic Drag Method
       [0]

     Normal Interfacial Drag Friction Factor
       [1000000]

     Tangential Interfacial Drag Friction Factor
       [10]

     Length scale
       [0.0001]
     ```

     The options for an `Anisotropic Drag Method` of `1` are as follows:

     ```
     Anisotropic Drag Method
       [0] 1

     Viscosity option
       [2]

     Normal Interfacial Drag Friction Factor
       [1000000]

     Tangential Interfacial Drag Friction Factor
       [10]

     Length scale
       [0.0001]
     ```

4. If you have enabled **Phase Localized Discretization** in the **Multiphase Model** dialog box, click the **Discretization** tab to enable the use of the **Phase Localized Compressive Scheme**. For information about applying the various schemes, refer to Discretizing Using the Phase Localized Compressive Scheme (p. 1443).

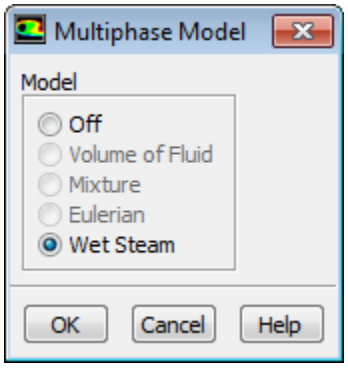
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.6. Setting Up the Wet Steam Model

After you have enabled the density-based solver in ANSYS Fluent, you can choose the wet steam model (see Wet Steam Model Theory in the Theory Guide) by opening the **Multiphase Model** dialog box and selecting the **Wet Steam** option.

Setup → Models → Multiphase → Edit...

Figure 24.63: The Multiphase Model Dialog Box with the Wet Steam Model Selected



This section includes information about using your own property functions and data with the wet steam model. Solution settings and strategies for the wet steam model can be found in Wet Steam Model (p. 1518). Postprocessing variables are described in Model-Specific Variables (p. 1520).

This section is organized as follows:

24.6.1. Using User-Defined Thermodynamic Wet Steam Properties
24.6.2. Writing the User-Defined Wet Steam Property Functions (UDWSPF)
24.6.3. Compiling Your UDWSPF and Building a Shared Library File
24.6.4. Loading the UDWSPF Shared Library File
24.6.5. UDWSPF Example

24.6.1. Using User-Defined Thermodynamic Wet Steam Properties

ANSYS Fluent allows you to use your own property functions and data with the wet steam model. This is achieved with user-defined wet steam property functions (UDWSPF).

These user-defined functions are written in the C programming language and there is a certain programming format that must be used so that you can build a successful library that can be loaded into the ANSYS Fluent code.

The following is the procedure for using the user-defined wet steam property functions (UDWSPF):

1. Define the wet steam equation of state and all related thermodynamic and transport property equations.
2. Create a C source code file that conforms to the format defined in this section.
3. Start ANSYS Fluent and set up your case file in the usual way.
4. Turn on the wet steam model.
5. Compile your UDWSPF C functions and build a shared library file using the text user interface.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`define` → `models` → `multiphase` → `wet-steam` → `compile-user-defined-wetsteam-functions`

6. Load your newly created UDWSPF library using the text user interface.

   `define` → `models` → `multiphase` → `wet-steam` → `load-unload-user-defined-wet-steam-library`

7. Run your calculation.

Important

Note that the UDWSPF can only be used when the wet steam model is selected. Therefore, the UDWSPF are available for use with the density-based solver only.

24.6.2. Writing the User-Defined Wet Steam Property Functions (UDWSPF)

Creating a UDWSPF C function library is reasonably straightforward:

- The code must contain the `udf.h` file inclusion directive at the beginning of the source code. This allows the definitions for `DEFINE` macros and other ANSYS Fluent functions to be accessible during the compilation process.
- The code must include at least one in the UDF's `DEFINE` functions (that is `DEFINE_ON_DEMAND`) to be able to use the compiled UDFs utility.
- Any values that are passed to the solver by the UDWSPF or returned by the solver to the UDWSPF are assumed to be in SI units.
- You must use the principle set of user-defined wet steam property functions in your UDWSPF library, as described in the list that follows. These functions are the mechanism by which your thermodynamic property data is transferred to the ANSYS Fluent solver.

The following lists the user-defined wet steam property function names and arguments, as well as a short description of their functions. Function inputs from the ANSYS Fluent solver consist of one or more of the following variables: T = temperature (K), P = pressure (Pa), and ρ = vapor-phase density (kg/m^3).

- `void wetst_init(Domain *domain)`

  This will be called when you load the UDWSPF. You use it to initialize wet steam model constants or your own model constants. It returns nothing.

- `real wetst_satP(real T)`

  This is the saturated pressure function, which takes on temperature in K and returns saturation pressure in Pa.

- `real wetst_satT(real P, real T)`

  This is the saturated temperature function, which takes on pressure in Pa and a starting guess temperature in K and returns saturation temperature in K.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `real wetst_eosP(real rho, real T)`

  This is the equation of state, which takes on vapor density in kg/m^3 and Temperature in K and returns pressure in Pa.

- `real wetst_eosRHO(real P, real T)`

  This is the equation of state, which takes on pressure in Pa and temperature in K and returns vapor density in kg/m^3.

- `real wetst_cpv(real T, real rho)`

  This is the vapor specific heat at constant pressure, which takes on temperature in K and vapor density in kg/m^3 and returns specific heat at constant pressure in J/kg/K.

- `real wetst_cvv(real T, real rho)`

  This is the vapor specific heat at constant volume, which takes on temperature in K and vapor density in kg/m^3 and returns specific heat at constant volume in J/kg/K.

- `real wetst_hv(real T,real rho)`

  This is the vapor specific enthalpy, which takes on temperature in K and vapor density in kg/m^3 and returns specific enthalpy in J/Kg.

- `real wetst_sv(real T, real rho)`

  This is the vapor specific entropy, which takes on temperature in K and vapor density in kg/m^3 and returns specific entropy in J/Kg/K.

- `real wetst_muv(real T, real rho)`

  This is the vapor dynamic viscosity, which takes on temperature in K and vapor density in kg/m^3 and returns viscosity in kg/m/s.

- `real wetst_ktv(real T, real rho)`

  This is the vapor thermal conductivity, which takes on temperature in K and vapor density in kg/m^3 and returns thermal conductivity in W/m/K.

- `real wetst_rhol(real T)`

  This is the saturated liquid density, which takes on temperature in K and returns liquid density in kg/m^3.

- `real wetst_cpl(real T)`

  This is the saturated liquid specific heat at constant pressure, which takes on temperature in K and returns liquid specific heat in J/kg/K.

- `real wetst_mul(real T)`

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This is the liquid dynamic viscosity, which takes on Temperature in K and returns dynamic viscosity in kg/m/s.

- `real wetst_ktl(real T)`

  This is the liquid thermal conductivity, which takes on temperature in K and returns thermal conductivity in W/m/K.

- `real wetst_surft(real T)`

  This is the liquid surface tension, which takes on Temperature in K and returns surface tension N/m.

At the end of the code you must define a structure of type `WS_Functions` whose members are pointers to the principle functions listed previously. The structure is of type `WS_Functions` and its name is `WetSteamFunctionList`.

```
UDF_EXPORT  WS_Functions  WetSteamFunctionList  =
 {
     wetst_init,     /*initialization  function*/
     wetst_satP,     /*Saturation  pressure*/
     wetst_satT,     /*Saturation  temperature*/
     wetst_eosP,     /*equation  of  state*/
     wetst_eosRHO,   /*equation  of  state*/
     wetst_hv,       /*vapor  enthalpy*/
     wetst_sv,       /*vapor  entropy*/
     wetst_cpv,      /*vapor  isobaric  specific  heat*/
     wetst_cvv,      /*vapor  isochoric  specific  heat*/
     wetst_muv,      /*vapor  dynamic  viscosity*/
     wetst_ktv,      /*vapor  thermal  conductivity*/
     wetst_rhol,     /*sat.  liquid  density*/
     wetst_cpl,      /*sat.  liquid  specific  heat*/
     wetst_mul,      /*sat.  liquid  viscosity*/
     wetst_ktl,      /*sat.  liquid  thermal  conductivity*/
     wetst_surft     /*liquid  surface  tension*/
 };
```

24.6.3. Compiling Your UDWSPF and Building a Shared Library File

This section presents the steps you will need to follow to compile your UDWSPF C code and build a shared library file. This process requires the use of a C compiler. For more details on compiler requirements, see Compilers in the *Fluent Customization Manual*.

Important

To use the UDWSPF you will need to first build the UDWSPF library by compiling your UDWSPF C code and then loading the library into the ANSYS Fluent code.

The UDWSPF shared library is built in the same way that the ANSYS Fluent executable itself is built. Internally, a script called `Makefile` is used to invoke the system C compiler to build an object code library that contains the native machine language translation of your higher-level C source code. This shared library is then loaded into ANSYS Fluent (either at run time or automatically when a case file is read) by a process called *dynamic loading*. The object libraries are specific to the computer architecture being used, as well as to the particular version of the ANSYS Fluent executable being run. The libraries must, therefore, be rebuilt any time ANSYS Fluent is upgraded, when the computer's operating system level changes, or when the job is run on a different type of computer.

The general procedure for compiling UDWSPF C code is as follows:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Place the UDWSPF C code in your working directory (that is, where your case file resides).
2. Launch ANSYS Fluent.
3. Read your case file into ANSYS Fluent.
4. You can now compile your UDWSPF C code and build a shared library file using the commands provided in the text command interface (TUI):
   a. Select the `define/models/multiphase/wet-steam` menu item.

      `define` → `models` → `multiphase` → `wet-steam`

      If the Wet Steam model has not been enabled yet, use the following TUI commands to enable it:

      ```
      define/models/multiphase/wet-steam> enable?

      Enable wet steam model? [no] yes
      ```

   b. Select the `compile-user-defined-wetsteam-functions` option.
   c. Enter the compiled UDWSPF library name.

      The name given here is the name of the directory where the shared library (for example, `libudf`) will reside. For example, if you press **Enter** then a directory should exist with the name `libudf`, and this directory will contain library file called `libudf`. If, however, you type a new library name such as `mywetsteam`, then a directory called `mywetsteam` will be created and it will contain the library `libudf`.
   d. Continue on with the procedure when prompted.
   e. Enter the C source file names.

      Important

      Ideally you should place all of your functions into a single file. However, you can split them into separate files if desired.

   f. Enter the header file names, if applicable. If you do not have an extra header file, then press **Enter** when prompted.

      ANSYS Fluent will then start compiling the UDWSPF C code and put it in the appropriate architecture directory.

24.6.4. Loading the UDWSPF Shared Library File

To load the UDWSPF library, perform the following steps:

- Go to the `define/models/multiphase/wet-steam` menu item in the text user interface.

  `define` → `models` → `multiphase` → `wet-steam`

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select the `load-unload-user-defined-wetsteam-library` option and follow the procedure when prompted.

  If the loading of the UDWSPF library is successful, you will see a message similar to the following:

```
Opening user-defined wet steam library "libudf"...
Library "libudf/lnamd64/2d/libudf.so" opened

Setting material properties to Wet-Steam...

Initializing user defined material properties...
```

24.6.5. UDWSPF Example

This section describe a simple UDWSPF. You can use this example as the basis for your own UDWSPF code. For approximate calculations at low pressure, the simple ideal-gas equation of state and constant isobaric specific heat is assumed and used. The properties at the saturated liquid line and the saturated vapor line used in this example are similar to the one used by ANSYS Fluent.

```
/*********************************************************************/
 /* User Defined Wet Steam Properties:
   EOS    : Ideal Gas Eq.
   Vapor Sat. Line : W.C.Reynolds tables (1979)
   Liquid Sat. Line: E. Eckert & R. Drake book (1972)

   Use ideal-gas EOS with Steam properties
   to model wet steam condensation in low pressure nozzle
   Author: L. Zori
   Date : Jan. 29 2004
 */
 /*********************************************************************/
 #include "udf.h"
 #include "stdio.h"
 #include "ctype.h"
 #include "stdarg.h"

 /*Global Constants for this model*/
 real ws_TPP = 338.150 ;
 real ws_aaa = 0.01 ;
 real cpg = 1882.0;/* Cp-vapor at low-pressure region*/

 DEFINE_ON_DEMAND(I_do_nothing)
 {
    /* This is a dummy function to allow us to use */
    /* the Compiled UDFs utility      */
 }


 void wetst_init(Domain *domain)
 {
    /*
      You must initialize these material property constants.
      they will be used in the wet steam model in fluent
    */
    ws_Tc = 647.286  ;/*Critical Temp. */
    ws_Pc = 22089000.00 ;/*Critical Pressure */
    mw_f  = 18.016 ;/*fluid droplet molecular weight (water) */
    Rgas_v  = 461.50 ;/*vapor Gas Const*/
 }

 real wetst_satP(real T)
 {
    real psat;
    real SUM=0.0;
    real pratio;
    real F;
    real a1 = -7.41924200 ;
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
   real a2 = 2.97210000E-01;
   real a3 = -1.15528600E-01;
   real a4 = 8.68563500E-03;
   real a5 = 1.09409899E-03;
   real a6 = -4.39993000E-03;
   real a7 = 2.52065800E-03;
   real a8 = -5.21868400E-04;
   if (T > ws_Tc) T = ws_Tc ;
   F  = ws_aaa*(T - ws_TPP)  ;
   SUM = a1 + F*(a2+ F*(a3+ F*(a4+ F*(a5+ F*(a6+ F*(a7+ F*a8))))));
   pratio = (ws_Tc/T - 1.0)*SUM;
   psat  = ws_Pc *exp(pratio) ;
  return psat; /*Pa */
}


real
wetst_satT(real P, real T)
{
   real tsat;
   real dT, dTA,dTM,dP,p1,p2,dPdT;
  real dt = 1.e-4;
   int i ;
     for (i=0; i<25; ++i)
      {
        if (T > ws_Tc) T = ws_Tc-0.5;

        p1= wetst_satP(T)  ;
        p2= wetst_satP(T+dt) ;
        dPdT = (p2-p1)/dt;

        dP = P - p1 ;

        dT = dP/dPdT  ;

        dTA = fabs(dT);
        T = T + dT;
        if (fabs(dT)<TEMP_eps*T) break;
        }
     tsat = T;
   return tsat; /*K */
}

real
wetst_eosP(real rho, real T)
{
   real P;

   P = rho* Rgas_v * T ;

  return P; /*Pa */
}


real
wetst_eosRHO(real P, real T)
{
    real rho;

    rho = P/(Rgas_v * T) ;

    return rho; /*kg/m3 */
}


real
wetst_cpv(real T, real rho)
{
    real cp;

    cp = cpg ;

    return cp; /* (J/Kg/K) */
```

```
}

real
wetst_cvv(real T, real rho)
{
   real cv;

   cv = wetst_cpv(T,rho) - Rgas_v ;

   return cv; /* (J/Kg/K) */
}

real
wetst_hv(real T,real rho)
{
   real h;

   h = T* wetst_cpv(T,rho) ;

  return h; /* (J/Kg) */
}



real
wetst_sv(real T, real rho)
{
   real s ;

   real TDatum=288.15;
   real PDatum=1.01325e5;
   s=wetst_cpv(T,rho)*log(T/TDatum)+
   Rgas_v*log(PDatum/(Rgas_v*T*rho));

 return s; /* (J/Kg/K) */
}


real
wetst_muv(real T, real rho)
{
   real muv;

   muv=1.7894e-05 ;

   return muv; /* (Kg/m/s) */
}

real
wetst_ktv(real T, real rho)
{
   real ktv;

   ktv=0.0242 ;

   return ktv; /* W/m/K */
}


real
wetst_rhol(real T)
{
   real rhol;

   real SUM = 0.0;
   int ii;
   int i;
   real rhoc = 317.0;
   real D[8];

   D[0] =  3.6711257 ;
   D[1] = -2.8512396E+01 ;
   D[2] =  2.2265240E+02 ;
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
   D[3] = -8.8243852E+02 ;
   D[4] =  2.0002765E+03 ;
   D[5] = -2.6122557E+03 ;
   D[6] =  1.8297674E+03 ;
   D[7] = -5.3350520E+02 ;
   if (T > ws_Tc) T = ws_Tc ;
   for(ii=0;ii 8;++ii)
     {
        i = ii+1 ;
        SUM += D[ii] * pow((1.0 - T/ws_Tc), i/3.0);
     }
   rhol = rhoc*(1.0+SUM);

   return rhol; /* (Kg/m3) */
}


real
wetst_cpl(real T)
{
   real cpl;

  real a1= -36571.6 ;
   real a2= 555.217 ;
   real a3= -2.96724 ;
   real a4= 0.00778551;
   real a5= -1.00561e-05;
   real a6= 5.14336E-09;

   if (T > ws_Tc) T = ws_Tc ;
   cpl = a1 + T*(a2+ T*(a3+ T*(a4+ T*(a5+ T*a6)))) ;

   return cpl; /* (J/Kg/K) */
}


real
wetst_mul(real T)
{
   real mul;

   real a1= 0.530784;
   real a2= -0.00729561;
   real a3= 4.16604E-05 ;
   real a4= -1.26258E-07;
   real a5= 2.13969E-10;
   real a6= -1.92145E-13;
   real a7= 7.14092E-17;

   if (T > ws_Tc) T = ws_Tc ;
   mul = a1 + T*(a2+ T*(a3+ T*(a4+ T*(a5+ T*(a6+ T*a7))))) ;

   return mul; /* (Kg/m/s) */
}

real
wetst_ktl(real T)
{
 real ktl;

   real a1= -1.17633;
   real a2= 0.00791645;  real a3= 1.48603E-05;
   real a4= -1.31689E-07;
   real a5= 2.47590E-10;
   real a6= -1.55638E-13;

   if (T > ws_Tc) T = ws_Tc ;
   ktl = a1 + T*(a2+ T*(a3+ T*(a4+ T*(a5+ T*a6)))) ;

 return ktl; /* W/m/K */
}
```

```
real
wetst_surft(real T)
{
   real sigma;

   real Tr;
   real a1= 82.27  ;
   real a2= 75.612;
   real a3= -256.889 ;
   real a4= 95.928;

   if (T > ws_Tc) T = ws_Tc ;
   Tr  = T/ws_Tc ;
   sigma = 0.001*(a1 + Tr*(a2+ Tr*(a3+ Tr*a4))) ;

   return sigma ;/* N/m */
}

/* do not change the order of the function list */
UDF_EXPORT WS_Functions WetSteamFunctionList =
  {
  wetst_init,     /*initialization  function*/
  wetst_satP,     /*Saturation  pressure*/
  wetst_satT,     /*Saturation  temperature*/
  wetst_eosP,     /*equation  of  state*/
  wetst_eosRHO,   /*equation  of  state*/
  wetst_hv,       /*vapor  enthalpy*/
  wetst_sv,       /*vapor  entropy*/
  wetst_cpv,      /*vapor  isobaric  specific  heat*/
  wetst_cvv,      /*vapor  isochoric  specific  heat*/
  wetst_muv,      /*vapor  dynamic  viscosity*/
  wetst_ktv,      /*vapor  thermal  conductivity*/
  wetst_rhol,     /*sat.  liquid  density*/
  wetst_cpl,      /*sat.  liquid  specific  heat*/
  wetst_mul,      /*sat.  liquid  viscosity*/
  wetst_ktl,      /*sat.  liquid  thermal  conductivity*/
  wetst_surft     /*liquid  surface  tension*/
 };
/************************************************************************/
```

24.7. Solution Strategies for Multiphase Modeling

Important

Double Precision is recommended for all multiphase cases.

For additional information, see the following sections:

24.7.1. Coupled Solution for Eulerian Multiphase Flows
24.7.2. Coupled Solution for VOF and Mixture Multiphase Flows
24.7.3. Selecting the Pressure-Velocity Coupling Method
24.7.4. Controlling the Volume Fraction Coupled Solution
24.7.5. VOF Model
24.7.6. Mixture Model
24.7.7. Eulerian Model
24.7.8. Wet Steam Model

24.7.1. Coupled Solution for Eulerian Multiphase Flows

In multiphase flow, the phasic momentum equations, the shared pressure, and the phasic volume fraction equations are highly coupled. Traditionally, these equations have been solved in a segregated fashion using some variation of the SIMPLE algorithm to couple the shared pressure with the momentum

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

equations. This is attained by effectively transforming the total continuity into a shared pressure. The ANSYS Fluent **Phase Coupled SIMPLE** algorithm has been successfully implemented and solves a wide range of multiphase flows. However, coupling the linearized system of equations in an implicit manner would offer a more robust alternative to the segregated approach.

One of the fundamental problems is that the resulting matrix is not symmetric and that the continuity constraint may contribute to a zero diagonal block, making the solution difficult to obtain. One way to circumvent this problem is to use direct solvers, but these are too expensive for large industrial cases. In addition, we need to avoid a zero diagonal, resulting from the continuity constraint, and like the segregated solver, we need to construct a pressure correction equation. In multiphase, we also have the additional problem of the vanishing phase, which for the coupled solver is important to ensure some continuity in the coefficients. Like the **Phase Coupled**, we use a Rhie and Chow type of scheme to calculate volume fluxes and to provide proper coupling between velocity and pressure, thereby avoiding unphysical oscillations.

Consider a single-phase system and let us denote the velocity correction components in the three Cartesian directions by u', v', and w' with p' denoting the shared pressure correction. These are discrete variables and can be expressed in the form p', U'. The linear system that is generated by the single-phase coupled solver is of the form

$$\begin{pmatrix} A_p & C_U \\ B_U & A_U \end{pmatrix} \begin{pmatrix} p' \\ U' \end{pmatrix} = \begin{pmatrix} S_p \\ S_U \end{pmatrix} \tag{24.16}$$

For a notation in component form $\left(p', u', v', w'\right)$

$$\begin{pmatrix} A_{pp} & C_u & C_v & C_w \\ B_u & A_{uu} & A_{uv} & A_{uw} \\ B_v & A_{vu} & A_{vv} & A_{vw} \\ B_w & A_{wu} & A_{wv} & A_{ww} \end{pmatrix} \begin{pmatrix} p' \\ u' \\ v' \\ w' \end{pmatrix} = \begin{pmatrix} S_p \\ S_u \\ S_v \\ S_w \end{pmatrix} \tag{24.17}$$

Now let us consider a multiphase system of n-phases and denote the phasic velocity correction components in the three Cartesian directions by u_k', v_k', and w_k' where the subscript k represents the phase notation, p' denotes the shared pressure correction and α_k' denotes the volume fraction correction (ANSYS Fluent can solve in both correction form for velocity and volume fraction and non-correction form). For simplicity the matrix will be shown for two phases. The vector solution is of the form $\left(p', u_1', v_1', w_1', u_2', v_2', w_2', \alpha_2'\right)$ or in a shorter notation $\left(p', U_1', U_2', \alpha_2'\right)$. The linear system would be an extension of the one generated by the coupled solver shown by Equation 24.16 (p. 1505).

$$\begin{pmatrix} A_p & C_{U1} & C_{U2} & D_{\alpha 2} \\ B_{U1} & A_{U1} & A_{U12} & D_{U1} \\ B_{U2} & A_{U21} & A_{U2} & D_{U2} \\ E_p & E_{U1} & E_{U2} & A_{\alpha 2} \end{pmatrix} \begin{pmatrix} p' \\ U_1' \\ U_2' \\ \alpha_2' \end{pmatrix} = \begin{pmatrix} S_p \\ S_{U1} \\ S_{U2} \\ S_{\alpha 2} \end{pmatrix} \tag{24.18}$$

This system can be easily generalized to n phases. The components of this matrix are also matrices.

For large problems we need to resort to iterative solvers. The ANSYS Fluent AMG Coupled solver with an ILU smoother has proved to be a robust method. Most coupled solvers also need a pseudo stepping method, adding more diagonal dominance to the matrix. Our method here is to use under-relaxation factors for momentum, which is equivalent to time stepping in steady flows. Similar to that of the single phase, we have introduced a steady Courant Number instead of an under-relaxation for velocities. Having this control is important when using second order numerical schemes in the convective terms.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For the sake of simplicity, input parameters for the **Coupled** solver are similar to the single-phase solver. We have the options for solving the whole system including volume fraction, or to treat the volume fraction solution in a segregated manner while preserving the pressure-velocity coupling for all phases.

Important

Equations in multiphase are more strongly linked than single phase and generally may need more under-relaxation, hence using the same values as single phase may not be ideal. A low Courant number would stabilize the solution.

See Selecting the Pressure-Velocity Coupling Method (p. 1507) for information about applying the various algorithms.

24.7.2. Coupled Solution for VOF and Mixture Multiphase Flows

We have the option of solving the multiphase system for VOF and mixture multiphase models in the following ways:

- Solving the continuity and momentum equations in a coupled manner (see Coupled Algorithm in the Theory Guide) and solving the volume fraction equation in a segregated manner.
- Solving the volume fraction equation in a coupled manner along with the continuity and momentum equations.

Solving the volume fraction equation in a coupled manner requires discretization of the volume fraction equation in the correction form along with the discretization of the continuity and momentum equation, as discussed in Coupled Algorithm in the Theory Guide.

The volume fraction equation could be represented in the discretized form as

$$\sum_k \sum_j a_{ij}^{\alpha u_k} u_{kj} + \sum_j a_{ij}^{\alpha p} p_j + \sum_j a_{ij}^{\alpha\alpha} \alpha_j = b_i^{\alpha} \tag{24.19}$$

The overall system of equations after being transformed to the correction form could be represented as

$$\sum_j [A]_{ij} \vec{X}_j = \vec{B}_i \tag{24.20}$$

For a 2D case and two-phase flow, the system could be expanded as follows:

$$A_{ij} = \begin{bmatrix} a_{ij}^{pp} & a_{ij}^{pu} & a_{ij}^{pv} & a_{ij}^{p\alpha} \\ a_{ij}^{up} & a_{ij}^{uu} & a_{ij}^{uv} & a_{ij}^{u\alpha} \\ a_{ij}^{vp} & a_{ij}^{vu} & a_{ij}^{vv} & a_{ij}^{v\alpha} \\ a_{ij}^{\alpha p} & a_{ij}^{\alpha u} & a_{ij}^{\alpha v} & a_{ij}^{\alpha\alpha} \end{bmatrix} \tag{24.21}$$

and the unknown and residual vectors have the form

$$\vec{X}_j = \begin{bmatrix} p'_i \\ u'_i \\ v'_i \\ \alpha'_i \end{bmatrix} \tag{24.22}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\vec{B}_i = \begin{bmatrix} -r_i^p \\ -r_i^u \\ -r_i^v \\ -r_i^\alpha \end{bmatrix} \tag{24.23}$$

where

A_{ij} = coefficient matrix

$\vec{X}_j$ = solution vector

$\vec{B}_i$ = residual vector

p' = pressure correction

u', v' = velocity corrections

α' = volume fraction correction

24.7.3. Selecting the Pressure-Velocity Coupling Method

The settings that are available in the **Solution Methods** task page (see Figure 24.64: The Solution Methods Task Page Displaying The Pressure-Velocity Coupling Options (p. 1509)) for solving the coupled system of equations arising in multiphase flows are:

Schemes

- **Phase Coupled SIMPLE** (Eulerian multiphase)
- **PISO** (VOF and mixture)
- **SIMPLE** (VOF and mixture)
- **SIMPLEC** (VOF and mixture)
- **Coupled** (all multiphase models)

Options

- **Coupled with Volume Fractions** (all multiphase models except drift-flux)
- **Solve N-Phase Volume Fraction Equations** (mixture and Eulerian models)

The **PISO**, **SIMPLE**, and **SIMPLEC** schemes apply to the VOF and mixture models and are discussed in Pressure-Velocity Coupling in the Theory Guide.

The **Phase Coupled SIMPLE** (PC-SIMPLE) is an extension of the SIMPLE algorithm [80] (p. 2891) to multiphase flows. The velocities are solved coupled by phases in a segregated fashion. Fluxes are reconstructed at the faces of the control volume and then a pressure correction equation is built based on total continuity. The coefficients of the pressure correction equations come from the coupled per phase momentum equations. This method has proven to be robust and it is the only method available for all previous versions of ANSYS Fluent.

The **Coupled** scheme (also known as **Multiphase Coupled** in previous ANSYS Fluent versions) solves all equations for phase velocity corrections and shared pressure correction simultaneously [34] (p. 2888). These methods incorporate the lift forces and the mass transfer terms implicitly into the general matrix.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This method works very efficiently in steady-state situations, or for transient problems when larger time steps are required.

The **Coupled with Volume Fractions** option (also known as **Full Multiphase Coupled** in previous ANSYS Fluent versions) couples velocity corrections, shared pressure corrections, and the correction for volume fraction simultaneously. Theoretically, it should be more efficient, however it may have some drawbacks in robustness and CPU time usage. The robustness issue stems from the lack of control of the solution of the volume fraction equation. The continuity constraint (sum of all volume fractions equals 1, and individual values limited between zero and one) cannot be enforced exactly during inner solver iterations, and slight variations from the physical limits may lead to divergence. Research is on-going in this area to improve the method. The method is advantageous for heterogeneous mass transfer when a low Courant number is given; it also works well in dilute situations.

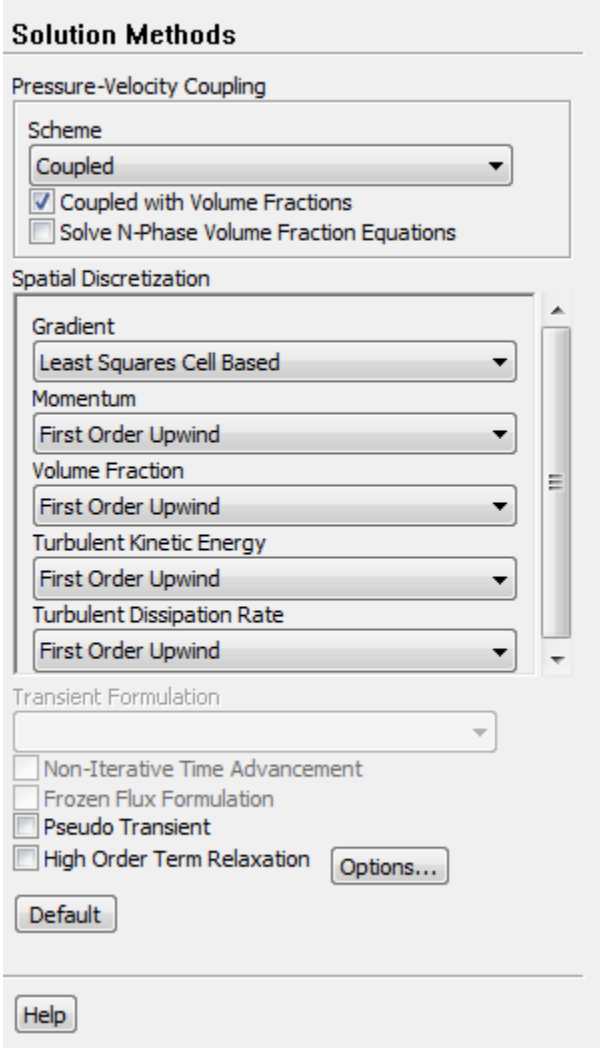
The Volume Fraction Coupling Method aims to achieve a faster steady-state solution compared to the segregated method of solving equations. It may not be a suitable option for transient applications due to the significant overhead in CPU time compared to the segregated method, unless it is run with a larger time step size.

Note

The **Coupled with Volume Fractions** option is available in the interface after you have selected **Coupled** from the **Scheme** drop-down list for **Pressure-Velocity Coupling**. For steady-state cases, the **Pseudo Transient** option will be enabled automatically when you enable the **Coupled with Volume Fractions** option for the VOF and mixture models.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.64: The Solution Methods Task Page Displaying The Pressure-Velocity Coupling Options



24.7.3.1. Limitations and Recommendations of the Coupled with Volume Fraction Options for the VOF and Mixture Models

The coupled with volume fractions option has the following limitations:

- It is not available when **Slip Velocity** is enabled for the **Mixture** multiphase model.
- It is not supported when using the Singhal-Et-Al cavitation model.
- It is not supported when the **Explicit** formulation for volume fraction is selected.

Recommended uses of the coupled with volume fractions option:

- The Pseudo-transient solver is recommended for steady-state calculations.
- It is recommended that you use lower under-relaxation factors for momentum for higher order schemes.
- For marine applications, it is recommended that you use a low-order variant or hybrid treatment for the Rhie-chow face flux interpolation (see High-Order Rhie-Chow Face Flux Interpolation (p. 1514)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- It is recommended that you use the expert text command options `solve/set/coupled-vof-expert` for better stability when using the VOF model (see `solve/` in the Text Command List).

24.7.3.2. Solving N-Phase Volume Fraction Equations

For a multiphase flow, the sum of the phase volume fractions must always equal 1. By default, Fluent enforces this constraint by solving the volume fraction equations for each of the secondary phases and then setting the volume fraction of the primary phase to the complement.

In certain cases, where poor convergence and/or mass imbalance are observed, enabling **Solve N-Phase Volume Fraction Equations** may improve solutions, although in practice the results obtained from using this option are case dependent. If selected, this option will solve all volume fraction equations, including both primary and secondary phases. The resulting phase volume fractions are then scaled in order to satisfy the requirement of summing to 1. This approach is more computationally expensive than the default approach and, in general, should not be necessary for the simulation.

Note

The **Solve N-Phase Volume Fraction Equations** option is not available for the **Volume of Fluid** model or **Eulerian** with **Multi-Fluid VOF Model** enabled.

24.7.4. Controlling the Volume Fraction Coupled Solution

When using the **Coupled with Volume Fractions** scheme, you will need to specify the following in the **Solution Controls** task page (see Figure 24.65: The Solution Controls Task Page Displaying the Coupled Volume Fraction Method for the VOF and Mixture Models (p. 1511)):

- **VOF and Mixture Multiphase Model:**
  - For steady-state cases using the pseudo transient solver, specify the **Volume Fraction Courant Number**, and the **Pseudo Transient Explicit Relaxation Factors**. The **Pseudo Transient Explicit Relaxation Factors** are described in Setting Pseudo Transient Explicit Relaxation Factors (p. 1602).

    Note

    The **Pseudo Transient Explicit Relaxation Factors** for the **Volume Fraction** is set to 0.5 by default.

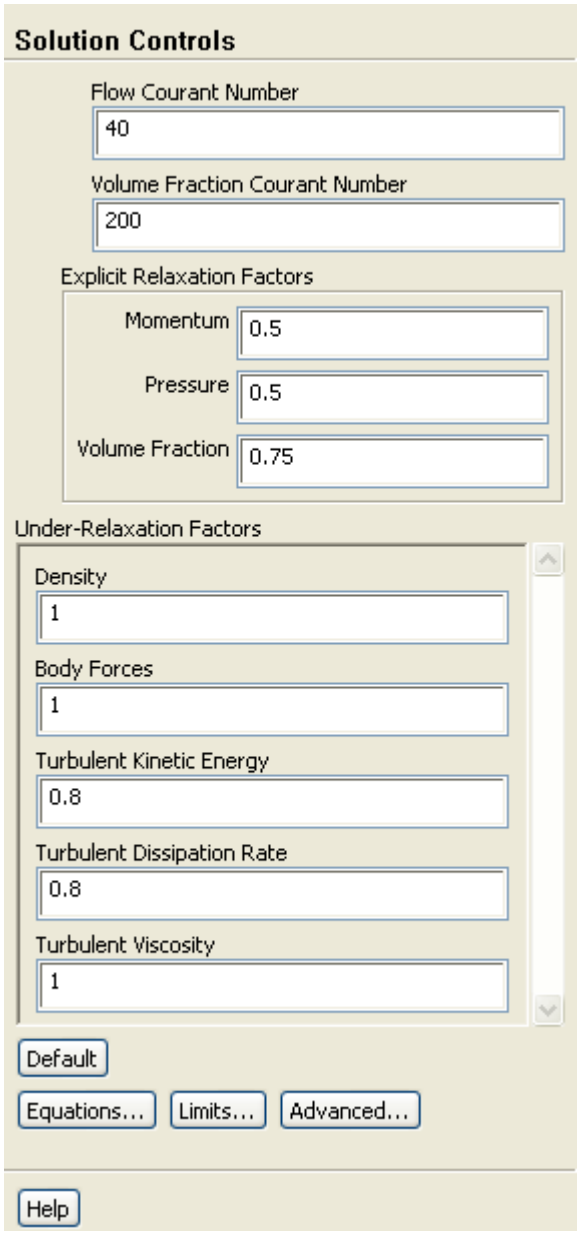
  - For transient cases or steady-state cases not involving the pseudo transient solver, enter the **Flow Courant Number**, the **Volume Fraction Courant Number**, **Explicit Relaxation Factors**, and the **Under-Relaxation Factors**.

    Note

    The **Volume Fraction** is set as an **Explicit Relaxation Factor** and is 0.75 by default.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.65: The Solution Controls Task Page Displaying the Coupled Volume Fraction Method for the VOF and Mixture Models



- **Eulerian Multiphase Model:**

  – For steady-state cases using the pseudo transient solver, specify the **Pseudo Transient Explicit Relaxation Factors** as described in Setting Pseudo Transient Explicit Relaxation Factors (p. 1602).

  Note

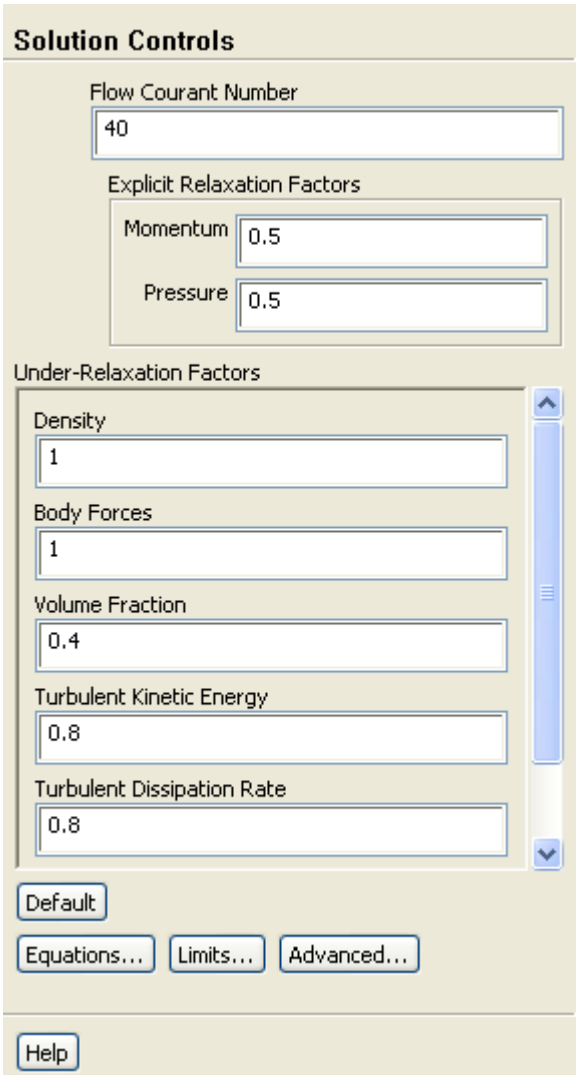
  The **Pseudo Transient Explicit Relaxation Factors** for the **Volume Fraction** is set to 0.5 by default.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– For transient cases or steady-state cases not involving the pseudo transient solver, enter the **Flow Courant Number**, the **Explicit Relaxation Factors**, and the **Under-Relaxation Factors**.

> **Note**
>
> The under-relaxation for **Volume Fraction** is set by using an implicit **Under-Relaxation Factor**, rather than a **Volume Fraction Courant Number**, and is 0.5 by default.

Figure 24.66: The Solution Controls Task Page Displaying the Coupled Volume Fraction Method for the Eulerian Multiphase Model



24.7.5. VOF Model

Several recommendations for improving the accuracy and convergence of the VOF solution are presented here.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.7.5.1. Setting the Reference Pressure Location

The site of the reference pressure can be moved to a location that will result in less round-off in the pressure calculation. By default, the reference pressure location is the center of the cell at or closest to the point (0,0,0). You can move this location by specifying a new **Reference Pressure Location** in the Operating Conditions Dialog Box (p. 2382).

Setup → Cell Zone Conditions → Operating Conditions...

The position that you choose should be in a region that will always contain the least dense of the fluids (for example, the gas phase, if you have a gas phase and one or more liquid phases). This is because variations in the static pressure are larger in a more dense fluid than in a less dense fluid, given the same velocity distribution. If the zero of the relative pressure field is in a region where the pressure variations are small, less round-off will occur than if the variations occur in a field of large nonzero values. Thus in systems containing air and water, for example, it is important that the reference pressure location be in the portion of the domain filled with air rather than that filled with water.

24.7.5.2. Pressure Interpolation Scheme

For all VOF calculations, you should use the body-force-weighted pressure interpolation scheme or the PRESTO! scheme.

Solution → Controls

24.7.5.3. Discretization Scheme Selection

The availability of discretization schemes for volume fraction is summarized in Spatial Discretization Schemes for Volume Fraction (p. 1373).

If you are modeling a sharp interface regime on meshes that have high aspect ratio cells, large jumps in cell size, or high skewness, interface capturing schemes such as **Compressive** and **CICSAM** will not produce very sharp results. For such cases, you should use the **Geo-Reconstruct** scheme.

For meshes of good quality, it is recommended that you use the **Compressive** scheme, which is available in both explicit and implicit formulations and is applicable to both sharp and dispersed flow regimes. The **Compressive** scheme also provides better accuracy compared with **CICSAM** and **Modified HRIC** for most cases. When used with the implicit formulation and the second order time scheme, the **Compressive** scheme produces quite a sharp interface.

When using the **Compressive** spatial discretization scheme, it is recommended that you use step-wise sharpening after the flow transitions from the diffused zone to the sharpening zone. In other words, you would want to transition from first order to second order, followed by a transition from second order to compressive. You would want to take this approach if the flow has a smooth transition from a sharp interfacial zone to a diffused interfacial zone. However, if the flow has a nonuniform transition from a diffused interfacial zone to a sharp interfacial zone, this might create unphysical sharpening of the interface if not handled properly, especially for transient cases. For example, transitioning from a slope limiter of 2 (compressive) to a slope limiter of 0 or 1 (first order or second order) might be acceptable, but a first order to compressive transition might create unphysical sharpening of the interface.

If you are solving a problem with the **evaporation-condensation** mass transfer mechanism enabled (Including Mass Transfer Effects (p. 1383)) it is recommended that you use one of the diffusive interface

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

capturing schemes such as **QUICK**, **HRIC**, or **Phase Localized Compressive** (Discretizing Using the Phase Localized Compressive Scheme (p. 1443)).

Important

The **BGM** scheme produces a sharp interface, which may result in poor convergence in some cases. In such situations, we recommend you use a low value for the VOF under-relaxation. In addition, you can start with the **Compressive** or **Modified HRIC** scheme and then switch to the **BGM** scheme.

24.7.5.4. High-Order Rhie-Chow Face Flux Interpolation

In VOF modeling, using a high-order discretization scheme for the momentum transport equations may reduce the stability of the solution compared to cases using first-order discretization. In such situations, there are a couple of recommendations:

1. Use a low-order variant of the Rhie-Chow face flux interpolation. This is enabled using the following text command:

   `solve` → `set` → `numerics`

   You will be asked

   ```
   disable high order Rhie-Chow flux? [no]
   ```

   to which you will respond `yes`.

2. Use a hybrid treatment of high-order Rhie-Chow face flux interpolation. This is enabled using the following text command:

   `solve` → `set` → `vof-numerics`

   You will be asked

   ```
   Use hybrid treatment for high order Rhie-Chow flux?  [no]
   ```

   to which you will respond `yes`.

   Hybrid treatment allows you to use a high-order variant of the Rhie-Chow face flux interpolation everywhere inside the domain, except in the vicinity of the interface where a low-order variant is used for the interpolation of the face flux. This treatment could be helpful to get better convergence without compromising much of the accuracy.

24.7.5.5. Treatment of Unsteady Terms in Rhie-Chow Face Flux Interpolation

For cases using Moving Mesh/Dynamic Mesh/Multiple Reference Frame models, ANSYS Fluent does not consider the unsteady terms for Rhie-Chow face flux interpolation by default. Accounting for the unsteady terms helps to achieve better solution stability, especially when using a lower time step size with the PRESTO scheme.

To enable forced treatment of unsteady terms in Rhie-Chow face flux interpolation, use the following text command:

`solve` → `set` → `vof-numerics`

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You will be asked

```
Use forced treatment of unsteady terms in Rhie-Chow flux? [no]
```

to which you will respond yes.

24.7.5.6. Using Unstructured Variant of PRESTO Pressure Scheme

For hexahedral and quadrilateral meshes, ANSYS Fluent uses the structured variant of the PRESTO scheme by default. For certain cases with non-conformal grid interfaces, the structured variant produces unexpected behavior at the intersection of the fluid-fluid and non-conformal grid interfaces. For these cases, using an unstructured variant of PRESTO provides better solution stability and avoids unexpectedly high velocities in the vicinity of the non-conformal grid interfaces.

To enable the unstructured variant of the PRESTO scheme, use the following text command:

`solve` → `set` → `vof-numerics`

You will be asked

```
Use unstructured variant of PRESTO pressure scheme? [no]
```

to which you will respond yes.

Note

The unstructured variant of PRESTO is automatically enabled for cases involving an unstructured mesh and this option is redundant in its usage.

24.7.5.7. Pressure-Velocity Coupling and Under-Relaxation for the Time-dependent Formulations

Another change that you should make to the solver settings is in the pressure-velocity coupling scheme and under-relaxation factors that you use. The PISO scheme is recommended for transient calculations in general. Using PISO allows for increased values on all under-relaxation factors, without a loss of solution stability. You can generally increase the under-relaxation factors for all variables to 1 and expect stability and a rapid rate of convergence (in the form of few iterations required per time step). For calculations on tetrahedral or triangular meshes, an under-relaxation factor of 0.7–0.8 for pressure is recommended for improved stability with the PISO scheme.

Solution → **Controls**

As with any ANSYS Fluent simulation, the under-relaxation factors will need to be decreased if the solution exhibits unstable, divergent behavior with the under-relaxation factors set to 1. Reducing the time step is another way to improve the stability.

24.7.5.8. Under-Relaxation for the Steady-State Formulation

If you are using the steady-state implicit VOF scheme, the under-relaxation factors for all variables should be set to values between 0.2 and 0.5 for improved stability.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.7.6. Mixture Model

24.7.6.1. Setting the Under-Relaxation Factor for the Slip Velocity

You should begin the mixture calculation with a low under-relaxation factor for the slip velocity. A value of 0.2 or less is recommended. If the solution shows good convergence behavior, you can increase this value gradually.

24.7.6.2. Calculating an Initial Solution

For some cases (for example, cyclone separation), you may be able to obtain a solution more quickly if you compute an initial solution without solving the volume fraction and slip velocity equations. Once you have set up the mixture model, you can temporarily disable these equations and compute an initial solution.

Solution → **Controls**

In the Equations Dialog Box (p. 2511), deselect **Volume Fraction** and **Slip Velocity** in the **Equations** list. You can then compute the initial flow field. Once a converged flow field is obtained, turn the **Volume Fraction** and **Slip Velocity** equations back on again, and compute the mixture solution.

24.7.7. Eulerian Model

24.7.7.1. Calculating an Initial Solution

To improve convergence behavior, you may want to compute an initial solution before solving the complete Eulerian multiphase model. There are multiple methods you can use to obtain an initial solution for an Eulerian multiphase calculation:

- Set up and solve the problem using the mixture model (with slip velocities) instead of the Eulerian model. You can then enable the Eulerian model, complete the setup, and continue the calculation using the mixture-model solution as a starting point.
- Set up the Eulerian multiphase calculation as usual, but compute the flow for only the primary phase. To do this, deselect **Volume Fraction** in the **Equations** list in the Equations Dialog Box (p. 2511). Once you have obtained an initial solution for the primary phase, turn the volume fraction equations back on and continue the calculation for all phases.
- Use the mass flow inlet boundary condition to initialize the flow conditions. It is recommended that you set the value of the volume fraction close to the value of the volume fraction at the inlet.
- At the beginning of the solution, a lower time step is recommended in order to reach convergence.
- If using the volume fraction explicit scheme, do not start with a large Courant number at the beginning of your run.
- If using the volume fraction explicit scheme, start a run with a lower time step and then increase the time step size. Alternatively, this could be done by using variable time stepping, which would increase the time step size based on the input parameters.
- For problems involving a free surface or sharp interfaces between the phases, it is recommended that you use the **symmetric** drag law, available in the Phase Interaction Dialog Box (p. 2363).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Variable time stepping is not recommended for compressible flows.

Important

You should *not* try to use a single-phase solution obtained without the mixture or Eulerian model as a starting point for an Eulerian multiphase calculation. Doing so will not improve convergence, and may make it even more difficult for the flow to converge.

24.7.7.2. Temporarily Ignoring Lift and Virtual Mass Forces

If you are planning to include the effects of lift and/or virtual mass forces in a steady-state Eulerian multiphase simulation, you can often reduce stability problems that sometimes occur in the early stages of the calculation by temporarily ignoring the action of the lift and the virtual mass forces. Once the solution without these forces starts to converge, you can interrupt the calculation, define these forces appropriately, and continue the calculation.

24.7.7.3. Using W-Cycle Multigrid

For problems involving a packed-bed granular phase with very small particle sizes (on the order of 10 μm), convergence can be obtained by using the W-cycle multigrid for the pressure. In the **Multigrid** tab, under **Fixed Cycle Parameters** in the Advanced Solution Controls Dialog Box (p. 2513), you may need to use higher values for **Pre-Sweeps**, **Post-Sweeps**, and **Max Cycles**. When you are choosing the values for these parameters, you should also increase the **Verbosity** to `1` in order to monitor the AMG performance; that is, to make sure that the pressure equation is solved to a desired level of convergence within the AMG solver during each global iteration. See Defining the Phases for the Eulerian Model (p. 1456) for more information about granular phases, and The V and W Cycles in the Theory Guide and Modifying Algebraic Multigrid Parameters (p. 1670) for details about multigrid cycles.

24.7.7.4. Including the Anisotropic Drag Law

When using the anisotropic drag law (Including the Multi-Fluid VOF Model (p. 1493)), it is recommended that you start the solution with a lower anisotropy ratio. After your solution has run for some time, you can then increase the ratio by reducing the friction factor in the tangential direction. Note that You can also start the solution with the symmetric drag law, then change to the anisotropic drag law.

Using a smaller under-relaxation for pressure and momentum may also help in convergence for cases with a higher anisotropy ratio.

If the flow for a particular phase is important in both directions (normal and tangential to the interface), use a lower anisotropy ratio, between 100-1000. A higher anisotropy ratio might cause an unstable solution for such cases. For a higher anisotropy ratio of more than 1000, a smaller under-relaxation for pressure and momentum is recommended. When using the coupled multiphase solver, if the solution is unstable with a higher anisotropy ratio, then reducing the courant number may be beneficial. `Anisotropic Drag Method` [1], with `Viscosity option` [2] is recommended for a higher viscosity ratio.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.7.8. Wet Steam Model

24.7.8.1. Boundary Conditions, Initialization, and Patching

When you use the wet steam model (described in Wet Steam Model Theory in the Theory Guide, and Setting Up the Wet Steam Model (p. 1495)), the following two field variables will show up in the inflow, outflow boundary dialog boxes, and in the **Solution Initialization** task page and **Patch** dialog boxes.

- **Liquid Mass Fraction** (or the wetness factor)

  In general, for dry steam entering flow boundaries the wetness factor is zero.

- **Log10 (Droplets Per Unit Volume)**

  In general this value is set to zero, indicating zero droplets entering the domain.

24.7.8.2. Solution Limits for the Wet Steam Model

When you select the wet steam model for the first time, a message is displayed indicating that the **Minimum Static Temperature** should be adjusted to 273 K since the accuracy of the built-in steam data is not guaranteed below a value of 273 K. If you use your own steam property functions, you can adjust this limit to whatever is permissible for your data.

To adjust the temperature limits, go to the **Solution Limits** dialog box.

Solution → **Controls** **Limits...**

The default maximum wetness factor or liquid mass fraction (β) is set to 0.1. In general, during the convergence process, it is common that this limit will be reached, but eventually the wetness factor will drop below the value of 0.1. However, in cases where the limit must be adjusted, you can do so using the text user interface.

`define` → `models` → `multiphase` → `wet-steam` → `set` → `max-liquid-mass-fraction`

Important

Note that the maximum wetness factor should not be set beyond 0.2 since the present model assumes a low wetness factor. When the wetness factor is greater than 0.1, the solution tends to be less stable due to the large source terms in the transport equations. Thus, the maximum wetness factor has been set to a default value of 0.1, which corresponds to the fact that most nozzle and turbine flows will have a wetness factor less than 0.1.

24.7.8.3. Solution Strategies for the Wet Steam Model

If you face convergence difficulties while solving wet steam flow, try to initially lower the CFL value and use first-order discretization schemes for the solution. If you are still unable to obtain a converged solution, then try the following solver settings:

1. Lower the under-relaxation factor for the wet steam equation below the current set value. The under-relaxation factor can be found in the **Solution Controls** task page.

   Solution → **Controls**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Solve for an initial solution with no condensation. Once you have obtained a proper initial solution, turn on the condensation.

   To turn condensation on or off, go to the **Solution Controls** task page.

   Solution → **Controls**

   In the **Equations** dialog box, deselect **Wet Steam** in the **Equations** list. When doing so, you are preventing condensation from taking place while still computing the flow based on steam properties. Once a converged flow field is obtained, turn the **Wet Steam** equation back on again and compute the mixture solution.

24.8. Multiphase Case Check

You can check the multiphase case setup using the multiphase check function available via the text user interface (TUI) command:

```
/report/mphase-summary
```

This function prints for you a summary of useful settings, important model information, and recommendations for choosing case parameters. Your case will be checked for compliance in the following categories:

- `MODELING`
- `MESHING`
- `PHASES`
- `PHASE INTERACTION`
- `BOUNDARY CONDITIONS`
- `CELL ZONE CONDITIONS`
- `OPERATING CONDITIONS`
- `SOLVER METHODS/CONTROLS`
- `ADVANCED SOLVER NUMERICS`
- `MULTIPHASE NUMERICS`
- `SOLUTION LIMITS`
- `MONITORS/RESIDUALS`
- `INITIALIZATION`
- `RUN CALCULATION`
- `POST PROCESSING`

Important

To execute this command, a case and valid data (solution or initialization) are required. If the case has been modified, you need to either reinitialize or rerun the case.

When prompted for `verbosity`, you can set the level of detail for the report. A value of **`0`** (the default) corresponds to a basic report, while a value of **`1`** corresponds to a more detailed report.

In each category, established rules and built-in best practices will be available, along with recommended changes to your current settings. At your discretion, you may choose to apply the recommendations, or keep your current settings. Refer to the appropriate sections in the Fluent User's Guide for more information about the recommended changes.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.9. Postprocessing for Multiphase Modeling

Each of the three general multiphase models provides a number of additional field functions that you can plot or report. You can also report flow rates for individual phases for all three models, and display velocity vectors for the individual phases in a mixture or Eulerian calculation.

Information about these postprocessing topics is provided in the following subsections:

24.9.1. Model-Specific Variables
24.9.2. Displaying Velocity Vectors
24.9.3. Reporting Fluxes
24.9.4. Reporting Forces on Walls
24.9.5. Reporting Flow Rates

24.9.1. Model-Specific Variables

When you use one of the general multiphase models, some additional field functions will be available for postprocessing, as listed in this section. Most field functions that are available in single phase calculations will be available for either the mixture or each individual phase, as appropriate for the general multiphase model and specific options that you are using. See Field Function Definitions (p. 1931) for a complete list of field functions and their definitions. Displaying Graphics (p. 1747) and Reporting Alphanumeric Data (p. 1885) explain how to generate graphics displays and reports of data.

24.9.1.1. VOF Model

For VOF calculations you can generate graphical plots or alphanumeric reports of the following additional item:

- **Volume fraction** (in the **Phases...** category)

  This item is available for each phase.

The variables that are not phase specific are available (for example, variables in the **Pressure...** and **Velocity...** categories) and represent mixture quantities. Thermal quantities will be available only for calculations that include the energy equation.

24.9.1.2. Mixture Model

For calculations with the mixture model, you can generate graphical plots or alphanumeric reports of the following additional items:

- **Diameter** (in the **Properties...** category)

  This item is available only for secondary phases.

- **Volume fraction** (in the **Phases...** category)

  This item is available only for secondary phases.

- **Interfacial Area Concentration** (in the **Interfacial Area Concentration...** category)

  This item is available only for secondary phases.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The variables that are not phase specific are available (for example, variables in the **Pressure...** category) represent mixture quantities. Thermal quantities will be available only for calculations that include the energy equation.

24.9.1.3. Eulerian Model

For Eulerian multiphase calculations you can generate graphical plots or alphanumeric reports of the following additional items:

- **Diameter** (in the **Properties...** category)

  This item is available only for secondary phases.

- **Granular Conductivity** (in the **Properties...** category)

  This item is available only for granular phases.

- **Granular Pressure** (in the **Granular Pressure...** category)

  This item is available only for granular phases.

- **Granular Temperature** (in the **Granular Temperature...** category)

  This item is available only for granular phases.

- **Volume fraction** (in the **Phases...** category)

  This item is available only for secondary phases.

- **Interfacial Area Concentration** (in the **Interfacial Area Concentration...** category)

  This item is available only for secondary phases.

The availability of turbulence quantities will depend on which multiphase turbulence model you used in the calculation. Thermal quantities will be available (on a per-phase basis) only for calculations that include the energy equation.

More advanced options for the mixture phase are available under the **Phases...** category, allowing you to select from a list of variables to postprocess. To access the entire list in the graphical user interface, type the following text command:

`solve` → `set` → `expert`

Retain most of the default settings, except when asked to `Keep temporary solver memory from being freed?`. Answering `yes` to this question will expose a list under the **Phases** category for the mixture phase, one of which will be the **Phase ID**. Selecting this option allows you to plot contours

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of phase IDs for the volume fraction, which will facilitate phase distribution display when more than two phases are present for free surface calculations.

Note

The `expert` option for not freeing temporary solver memory is incompatible with dynamic adaption in parallel.

Important

This option is available for all the multiphase models. However, note that only cell values should be plotted for this option. Make sure that the **Node Values** option is not selected as it will show the wrong phase ID contours at the interface.

24.9.1.4. Multiphase Species Transport

For calculations using species transport with either of the multiphase models, you can generate graphical plots or alphanumeric reports of the following additional items:

- **Mass Fraction of species-n** (in the **Species...** category)

  This item is available for each species.

- **Mole Fraction of species-n** (in the **Species...** category)

  This item is available for each species.

- **Molar Concentration of species-n** (in the **Species...** category)

  This item is available for each species.

- **Lam. Diff Coeff of species-n** (in the **Species...** category)

  This item is available for each species.

- **Eff. Diff. Coeff. of species-n** (in the **Species...** category)

  This item is available for each species.

- **Enthalpy of species-n** (in the **Species...** category)

  This item is available for each species.

- **Relative Humidity** (in the **Species...** category).

- **Turbulent Rate of Reaction-n** (in the **Reactions...** category)

  This item is available for each species.

- **Rate of Reaction** (in the **Reactions...** category).

- **Mass Transfer Rate n** (in the **Phase Interaction...** category)

  This item is available for each mass transfer mechanism that you defined.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Thermal quantities will be available only for calculations that include the energy equation.

24.9.1.5. Wet Steam Model

ANSYS Fluent provides a wide range of postprocessing information related to the wet steam model.

The wet steam related items can be found in **Wet Steam....** category of the variable selection drop-down list that appears in the postprocessing dialog boxes.

- **Liquid Mass Fraction**
- **Liquid Mass Generation Rate**
- **Log10 (Droplets Per Unit Volume)**
- **Log10 (Droplets Nucleation Rate)**
- **Steam Density (Gas-Phase)**
- **Liquid Density (Liquid-Phase)**
- **Mixture Density**
- **Saturation Ratio**
- **Saturation Pressure**
- **Saturation Temperature**
- **Subcooled Vapor Temperature**
- **Droplet Surface Tension**
- **Droplet Critical Radius (microns)**
- **Droplet Average Radius (microns)**
- **Droplet Growth Rate (microns/s)**

24.9.1.6. Dense Discrete Phase Model

For postprocessing, both the DPM and the Eulerian multiphase capabilities are retained. In addition to usual DPM postprocessing (Postprocessing for the Discrete Phase (p. 1329)), you can display, for example, vector plots of the particle's velocity field. Make sure to select the discrete phase from the **Phase** drop-down list. For transient simulations that include the dense discrete phase model, you can display the following when the **Unsteady Statistics...** category is selected:

- **Mean Velocity**
- **Mean Volume Fraction**
- **Mean Phase Diameter**
- **RMSE Velocity**
- **RMSE Volume Fraction**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **RMSE Phase Diameter**

Important

For the **Unsteady Statistics...** category to appear in the postprocessing dialog boxes, make sure that **Data Sampling for Time Statistics** is enabled in the **Run Calculation** task page, and that you have performed the calculation.

24.9.2. Displaying Velocity Vectors

For mixture and Eulerian calculations, it is possible to display velocity vectors for the individual phases using the **Vectors** dialog box.

Results → **Graphics** → **Vectors** **Edit...**

To display the velocity of a particular phase, select **Velocity** in the **Vectors of** drop-down list, and then select the desired phase in the **Phase** drop-down list. You can also choose **Relative Velocity** to display the phase velocity relative to a moving reference frame. To display the mixture velocity $\vec{v}_m$ (relevant for mixture model calculations only), select **Velocity** (or **Relative Velocity** for the mixture velocity relative to a moving reference frame), and **mixture** as the **Phase**. Note that you can color vectors by values of any available variable, for any phase you defined. To do so, make the appropriate selections in the **Color by** and following **Phase** drop-down lists.

24.9.3. Reporting Fluxes

When you use the **Flux Reports** dialog box to compute fluxes through boundaries, you will be able to specify whether the report is for the mixture or for an individual phase.

Results → **Reports** → **Fluxes** **Edit...**

Select **mixture** in the **Phase** drop-down list at the bottom of the dialog box to report fluxes for the mixture, or select the name of a phase to report fluxes just for that phase.

24.9.4. Reporting Forces on Walls

For Eulerian calculations, when you use the **Force Reports** dialog box to compute forces or moments on wall boundaries, you will be able to specify the individual phase for which you want to compute the forces.

Results → **Reports** → **Forces** **Edit...**

Select the name of the desired phase in the **Phase** drop-down list on the left side of the dialog box.

24.9.5. Reporting Flow Rates

You can obtain a report of mass flow rate for each phase (and the mixture) through each flow boundary using the `report/fluxes/mass-flow` text command:

`report` → `fluxes` → `mass-flow`

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you specify the phase of interest (the mixture or an individual phase), ANSYS Fluent will give you the option to list each zone, followed by a summary of the mass flow rate through that zone for the specified phase, or will summarize the mass flow rate for all zones. An example is shown below, demonstrating how to list the mass flow rate for all zones.

```
/report/fluxes> mf
(mixture water air)
domain id/name [mixture] air
all boundary/interior zones [yes]
Write to File? [no]

                            air
                  Mass Flow Rate               (kg/s)
------------------------------- --------------------
             spiral-press-outlet           -1.2330244
                 pressure-outlet           -9.7560663
                spiral-vel-inlet            0.6150589
                           walls                    0
                  velocity-inlet            4.9132133
                --------------- --------------------
                             Net           -5.4608185
```

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 25: Modeling Solidification and Melting

This chapter describes how you can model solidification and melting in ANSYS Fluent. For information about the theory behind the model, see Solidification and Melting in the Theory Guide. Information about using the model is organized into the following sections:

25.1. Setup Procedure
25.2. Procedures for Modeling Continuous Casting
25.3. Modeling Thermal and Solutal Buoyancy
25.4. Solution Procedure
25.5. Postprocessing

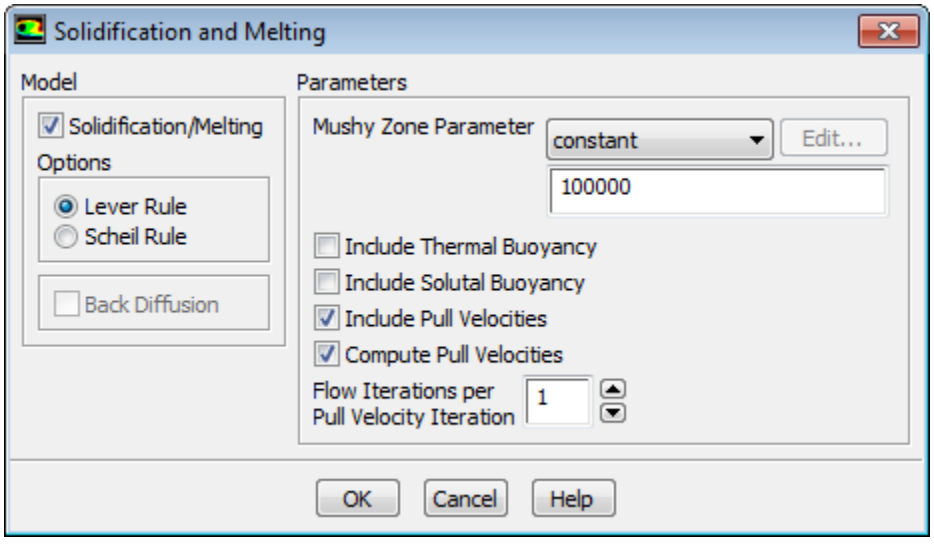
25.1. Setup Procedure

The procedure for setting up a solidification/melting problem is described below. (Note that this procedure includes only those steps necessary for the solidification/melting model itself; you will need to set up other models, boundary conditions, and so on, as usual.)

1. To use the solidification/melting model, enable the **Solidification/Melting** option in the **Solidification and Melting** dialog box (Figure 25.1: The Solidification and Melting Dialog Box (p. 1527)).

   Setup → **Models** → **Solidification & Melting** **Edit...**

   Figure 25.1: The Solidification and Melting Dialog Box

   ANSYS Fluent will automatically enable the energy equation, so you do not have to visit the **Energy** dialog box before turning on the solidification/melting model.

2. Under **Parameters**, specify the value of the **Mushy Zone Parameter** (A_{mush} in Equation 18.6) as a **constant**, or as a **user-defined** function. Refer to `DEFINE_SOLIDIFICATION_PARAMS` in the Fluent Customization Manual for detailed information about the user-defined function.

Values between 10^4 and 10^7 are recommended for most computations. The higher the value of the **Mushy Zone Parameter**, the steeper the damping curve becomes, and the faster the velocity drops to zero as the material solidifies. Very large values may cause the solution to oscillate as control volumes alternately solidify and melt with minor perturbations in liquid volume fraction.

3. If you want to include the pull velocity in your simulation (as described in Momentum Equations and Pull Velocity for Continuous Casting in the Theory Guide), enable the **Include Pull Velocities** option under **Parameters**.

4. If you are including pull velocities and you want ANSYS Fluent to compute them (using Equation 18.22) based on the specified velocity boundary conditions, as described in Pull Velocity for Continuous Casting in the Theory Guide, enable the **Compute Pull Velocities** option and specify the number of **Flow Iterations Per Pull Velocity Iteration**.

   Important

   It is not necessary to have ANSYS Fluent compute the pull velocities. See Procedures for Modeling Continuous Casting (p. 1530) for information about other approaches.

   The default value of 1 for the **Flow Iterations Per Pull Velocity Iteration** indicates that the pull velocity equations will be solved after each iteration of the solver. If you increase this value, the pull velocity equations will be solved less frequently. You may want to increase the number of **Flow Iterations Per Pull Velocity Iteration** if the liquid fraction equation is almost converged (that is, the position of the liquid-solid interface is not changing very much). This will speed up the calculation, although the residuals may jump when the pull velocities are updated.

5. Under **Options**, select either **Lever Rule** or **Scheil Rule**. See Species Equations in the Theory Guide for details.

   Important

   The **Lever Rule** and **Scheil Rule** options are available only when **Species Transport** is enabled in the **Species Model** dialog box.

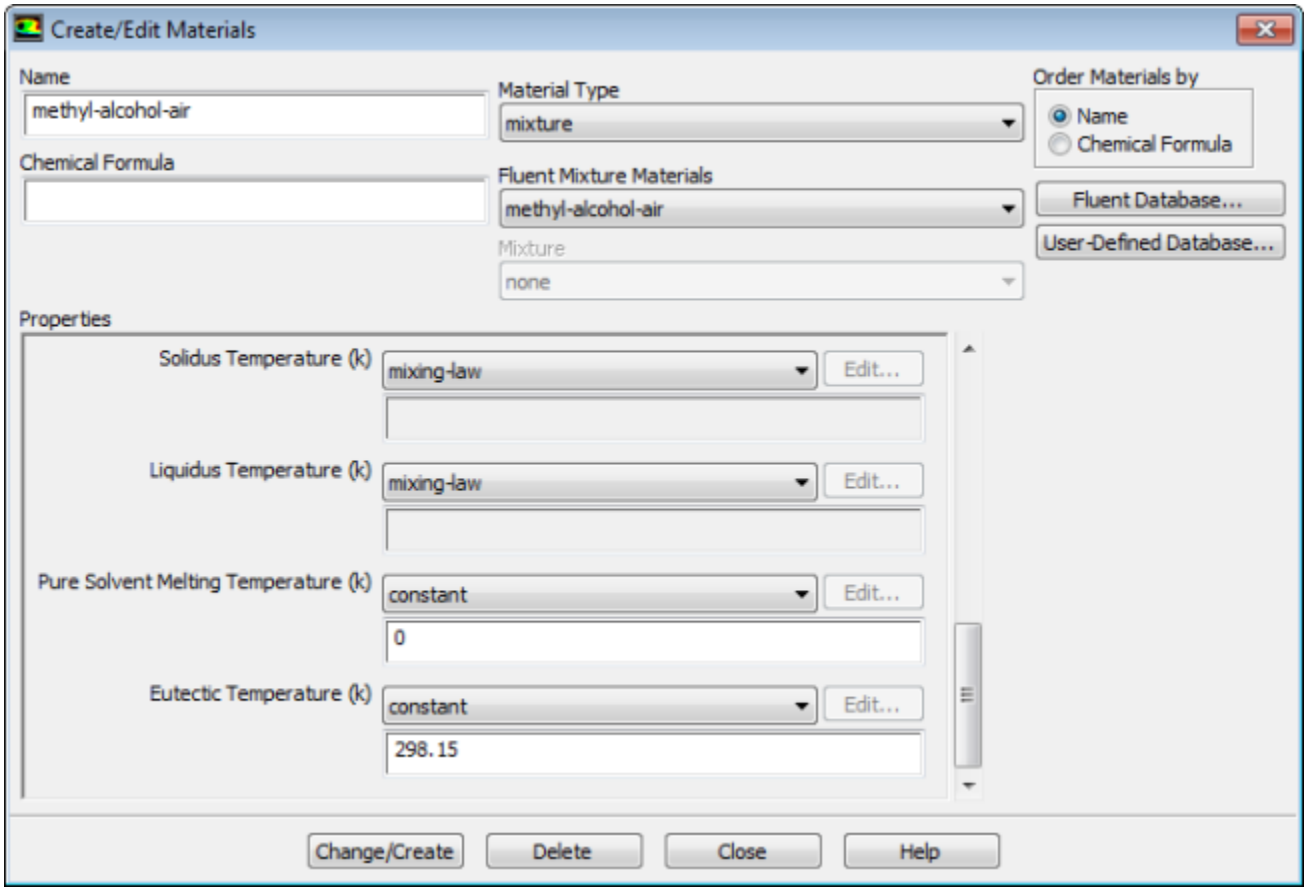
6. If you select **Scheil Rule**, then you can enable **Back Diffusion**. Enter either a **constant** or a **user-defined** function to specify the value of the **Back Diffusion Parameter** (γ in Equation 18.19). Refer to `DEFINE_SOLIDIFICATION_PARAMS` in the Fluent Customization Manual for detailed information about the user-defined function. Note that the value for the **Back Diffusion Parameter** must be between 0 and 1.

7. In the Create/Edit Materials Dialog Box (p. 2302) (Figure 25.2: The Create/Edit Materials Dialog Box for Melting and Solidification (p. 1529)), specify the **Pure Solvent Melting Heat** (L in Equation 18.4), **Solidus Temperature** ($T_{solidus}$ in Equation 18.3), and **Liquidus Temperature** ($T_{liquidus}$ in Equation 18.3) for the material being used in your model.

   Setup → **Materials**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 25.2: The Create/Edit Materials Dialog Box for Melting and Solidification



If you are solving for species transport, you need to specify properties for the mixture, including the method by which the **Solidus Temperature** and the **Liquidus Temperature** are calculated. The default method is the **mixing-law** (Equation 18.8 and Equation 18.9 in the Theory Guide), in which the solidus temperature and the liquidus temperature are calculated from the parameters provided for each solute (such as the slope of the liquidus line or partition coefficient). However, a **user-defined** function of type `DEFINE_PROPERTY` can be used to specify both of these temperatures. See the Fluent Customization Manual for examples of `DEFINE_PROPERTY`.

Important

It is highly recommended that you use the same method for specifying the **Solidus Temperature** and the **Liquidus Temperature**.

When defining the mixture, you will also specify the **Mass Diffusivity** ($D_{i,m,\text{liq}}$ in Equation 18.15 and Equation 18.18) and the **Eutectic Temperature** (T_{Eut} in Equation 18.10), as well as the **Pure Solvent Melting Heat** (L in Equation 18.4) and the **Pure Solvent Melting Temperature** (T_{melt} in Equation 18.8 and Equation 18.9). Note that the solvent is the last species listed under **Selected Species** in the **Species** dialog box.

For each solute, you have to specify the **Slope of Liquidus Line** (m_i in Equation 18.8 and Equation 18.9 in the Theory Guide) with respect to the concentration of the solute, the **Partition Coefficient** (K_i in Equation 18.8), the **Eutectic Mass Fraction** ($Y_{i,\text{Eut}}$ in Equation 18.10), and, if **Lever Rule** is selected

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in the **Solidification and Melting** dialog box, the coefficient for **Diffusion in Solid** ($D_{i,m,\text{sol}}$ in Equation 18.15). It is not necessary to specify m_i, K_i, $Y_{i,\text{Eut}}$, and $D_{i,m,\text{sol}}$ for the solvent.

8. Set the boundary conditions.

   Setup → **Boundary Conditions**

   In addition to the usual boundary conditions, consider the following:

   - If you want to account for the presence of an air gap between a wall and an adjacent solidified region (as described in Contact Resistance at Walls in the Theory Guide), specify a nonzero value, a profile, or a user-defined function for **Contact Resistance** (R_c in Equation 18.23) under **Thermal Conditions** in the Wall Dialog Box (p. 2456).

   - If you want to specify the gradient of the surface tension with respect to the temperature at a wall boundary, you can use the **Marangoni Stress** option for the wall **Shear Condition**. See Marangoni Stress (p. 363) for details.

   - If you want ANSYS Fluent to compute the pull velocities during the calculation, note how your specified velocity conditions are used in this calculation (see Pull Velocity for Continuous Casting in the Theory Guide).

Procedures for Modeling Continuous Casting (p. 1530) contains additional information about modeling continuous casting. See Solution Procedure (p. 1532) and Postprocessing (p. 1532) for information about solving a solidification/melting model and postprocessing the results.

25.2. Procedures for Modeling Continuous Casting

As described in Momentum Equations and Pull Velocity for Continuous Casting in the Theory Guide, you can include the pull velocities in your solidification/melting calculation to model continuous casting. There are three approaches to modeling continuous casting in ANSYS Fluent:

- Specify constant or variable pull velocities.

  To use this approach (the default), do not enable the **Compute Pull Velocities** option.

  If you use this approach, you will need to patch constant values or custom field functions for the pull velocities, after you initialize the solution.

  Solution → **Initialization** → **Patch...**

  See Patching Values in Selected Cells (p. 1593) for details about patching values. Note that it is acceptable to patch values for the pull velocities in the entire domain, because the patched values will be used only if the liquid fraction, β, is less than 1.

- Have ANSYS Fluent compute the pull velocities (using Equation 18.22) during the calculation, based on the specified velocity boundary conditions.

  To use this approach, enable the **Compute Pull Velocities** option. This method is computationally expensive, and is recommended only if the pull velocities are strongly dependent on the location of the liquid-solid interface.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you have ANSYS Fluent compute the pull velocities, then there are no additional inputs or setup procedures beyond those presented in Setup Procedure (p. 1527).

- Have ANSYS Fluent compute the pull velocities just once, and then use those values for the remainder of the calculation.

  To use this approach, perform one iteration with ANSYS Fluent computing the pull velocities, and then turn off the **Compute Pull Velocities** option and continue the calculation. For the remainder of the calculation, ANSYS Fluent will use the values computed for the pull velocities at the first iteration.

25.3. Modeling Thermal and Solutal Buoyancy

When the effects of thermal and solutal buoyancy are present, a flow can be induced inside the domain due to the effect of gravity on the variable density of the medium. In the case of multi-component solidification problems, the density variation takes place due to temperature changes and also due to species concentration gradients near the liquid-solid interface. The flow due to buoyancy with solidification and melting can be modeled in ANSYS Fluent using the thermal and solutal buoyancy options.

For more information on the theory behind buoyancy induced flow in solidification and melting problems, see Thermal and Solutal Buoyancy in the Theory Guide.

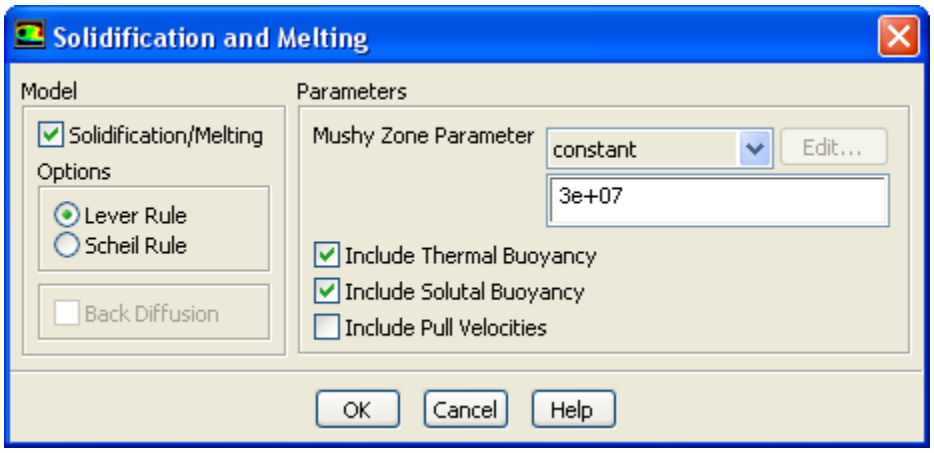
The procedure for setting up a solidification/melting problem is described in Setup Procedure (p. 1527). To include thermal and solutal buoyancy effects, perform the following steps:

1. Enable the **Include Thermal Buoyancy** and **Include Solutal Buoyancy** options in the **Solidification and Melting** dialog box (Figure 25.3: The Solidification and Melting Dialog Box (p. 1531)).

   Note

   The **Include Thermal Buoyancy** and **Include Solutal Buoyancy** options are available only when solidification is modeled with species transport.

   Figure 25.3: The Solidification and Melting Dialog Box



2. Define the operating conditions and properties for modeling thermal buoyancy as described in Natural Convection and Buoyancy-Driven Flows (p. 833).

3. To model solutal buoyancy, specify a value for the **Solutal Expansion Coefficient** for all the species except the last one in the mixture in the **Create/Edit Materials** dialog box.

Note

By default, the eutectic mass fraction of the solute is used as the reference species mass fraction of the solute for the calculation of the body force due to solutal buoyancy. Therefore, no additional input is required. However, in certain applications, it is not always desirable to use the default values of the reference mass fraction. For such cases, the solute mass fraction values can be entered through text user interface as follows:

```
 define/models/solidification-melting? yes
Include Thermal Buoyancy? yes
Include Solutal Buoyancy? yes
Use reference mass fraction of solutes? yes
Reference mass fraction of the species-i "value"
```

25.4. Solution Procedure

Before solving the coupled fluid flow and heat transfer problem, you may want to patch an initial temperature or solve the steady conduction problem as an initial condition. The coupled problem can then be solved as either steady or transient. Because of the nonlinear nature of these problems, however, in most cases a transient solution approach is preferred.

You can specify the under-relaxation factor applied to the liquid fraction equation in the Solution Controls Task Page (p. 2508).

Solution → **Controls**

Specify the desired value in the **Liquid Fraction Update** field under **Under-Relaxation Factors**. This sets the value of α_β in the following equation for updating the liquid fraction from one iteration (n) to the next ($n+1$):

$$\beta_{n+1}=\beta_n+\alpha_\beta\Delta\beta \tag{25.1}$$

where $\Delta\beta$ is the predicted change in liquid fraction.

In many cases, there is no need to change the default value of α_β. If, however, there are convergence difficulties, reducing the value may improve the solution convergence. Convergence difficulties can be expected in steady-state calculations, continuous casting simulations, simulations involving multicomponent solidification, and simulations where a large value of the mushy zone constant is used.

25.5. Postprocessing

For solidification/melting calculations, you can generate graphical plots or alphanumeric reports of the following items depending on which other models are enabled in the simulation. These quantities are available in the **Solidification/Melting...** category of the variable selection drop-down list that appears in postprocessing dialog boxes:

- **Liquid Fraction**
- **Contact Resistivity**

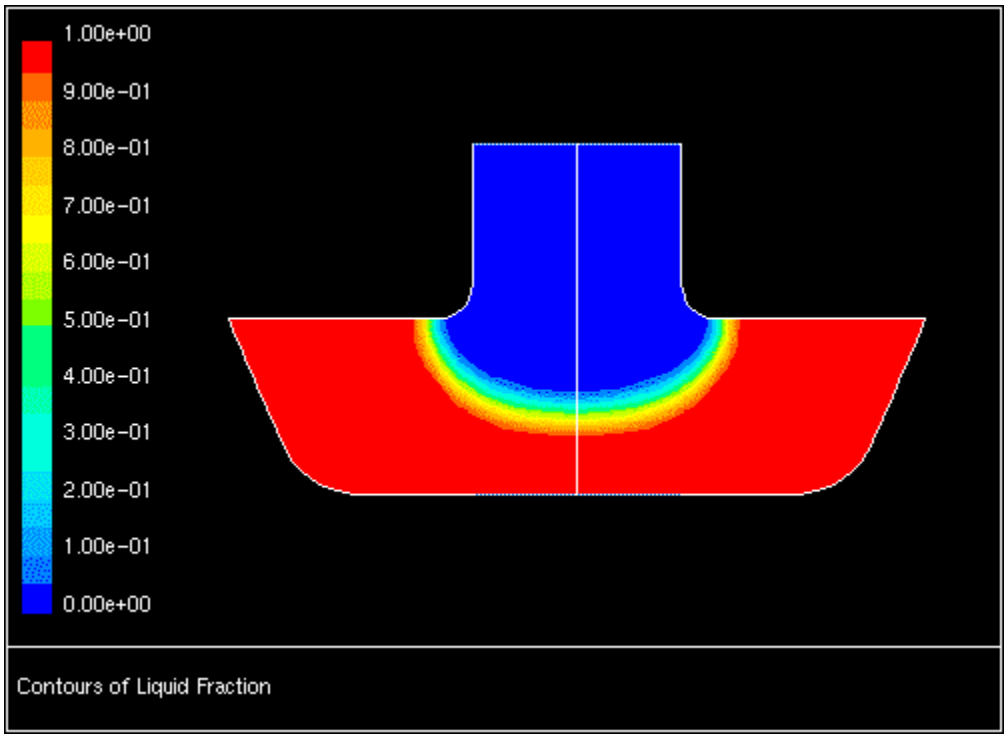
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Pull Velocity** (**X**, **Y**, **Z**, **Axial**, **Radial**, and **Swirl** components)
- **Liquidus Temperature**
- **Solidus Temperature**

The **Liquid Fraction** and **Contact Resistivity** solution variables are available for all solidification/melting simulations. The **Pull Velocity** components are available only if you are including pull velocities (either computed or specified) in the simulation. **Liquidus Temperature** and **Solidus Temperature** are available only if the **Species** model is set up to perform a multi-component solidification/melting simulation. See Field Function Definitions (p. 1931) for a complete list of field functions and their definitions. Displaying Graphics (p. 1747) and Reporting Alphanumeric Data (p. 1885) explain how to generate graphics displays and reports of data.

Figure 25.4: Liquid Fraction Contours for Continuous Crystal Growth (p. 1533) shows filled contours of liquid fraction for a continuous crystal growth simulation.

Figure 25.4: Liquid Fraction Contours for Continuous Crystal Growth



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 26: Modeling Eulerian Wall Films

The Eulerian Wall Film (EWF) model can be used to predict the creation and flow of thin liquid films on the surface of walls. This chapter presents information about the basic functionality of the Eulerian Wall Film (EWF) model. Additional information about the model is provided in the following sections:

26.1. Limitations
26.2. Setting Eulerian Wall Film Model Options
26.3. Setting Eulerian Wall Film Solution Controls
26.4. Setting Eulerian Wall Film Boundary, Initial, and Source Term Conditions
26.5. Postprocessing the Eulerian Wall Film

For more information about Eulerian Wall Film model theory, see Eulerian Wall Films in the Theory Guide. For more information about setting boundary conditions for liquid films at wall boundaries, see Wall Film Conditions for Walls (p. 381).

26.1. Limitations

The following limitations exist for the Eulerain Wall Film model:

- The Eulerian Wall Film model is available for 3D geometries only.
- Many models (for example, VOF multiphase flow or radiation) will not interact correctly with the film model without first modifying the boundary conditions using UDFs.
- Wallfilm behavior at junctions with baffles (zero-thickness walls) may lead to unphysical results due to the local mesh topology (wall, wall-shadow) and should be used with care, that is, isolated film walls must not share a common edge.
- The wall film can only be single-component, but not multi-component.
- The Eulerian Wall Film model is not compatible with non-conformal interfaces.
- The Eulerian Wall Film model is not compatible with mesh operations in Fluent (such as mesh adaption, cell separation, face zone extrusion, cell zone type change, and so on).

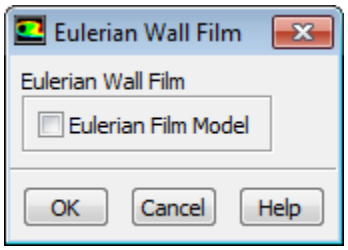
26.2. Setting Eulerian Wall Film Model Options

You can enable the Eulerian Wall Film model by selecting **Eulerian Wall Film** from the **Models** task page.

Setup → **Models** → **Eulerian Wall Film** **Edit...**

*This opens the **Eulerian Wall Film** dialog box.*

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



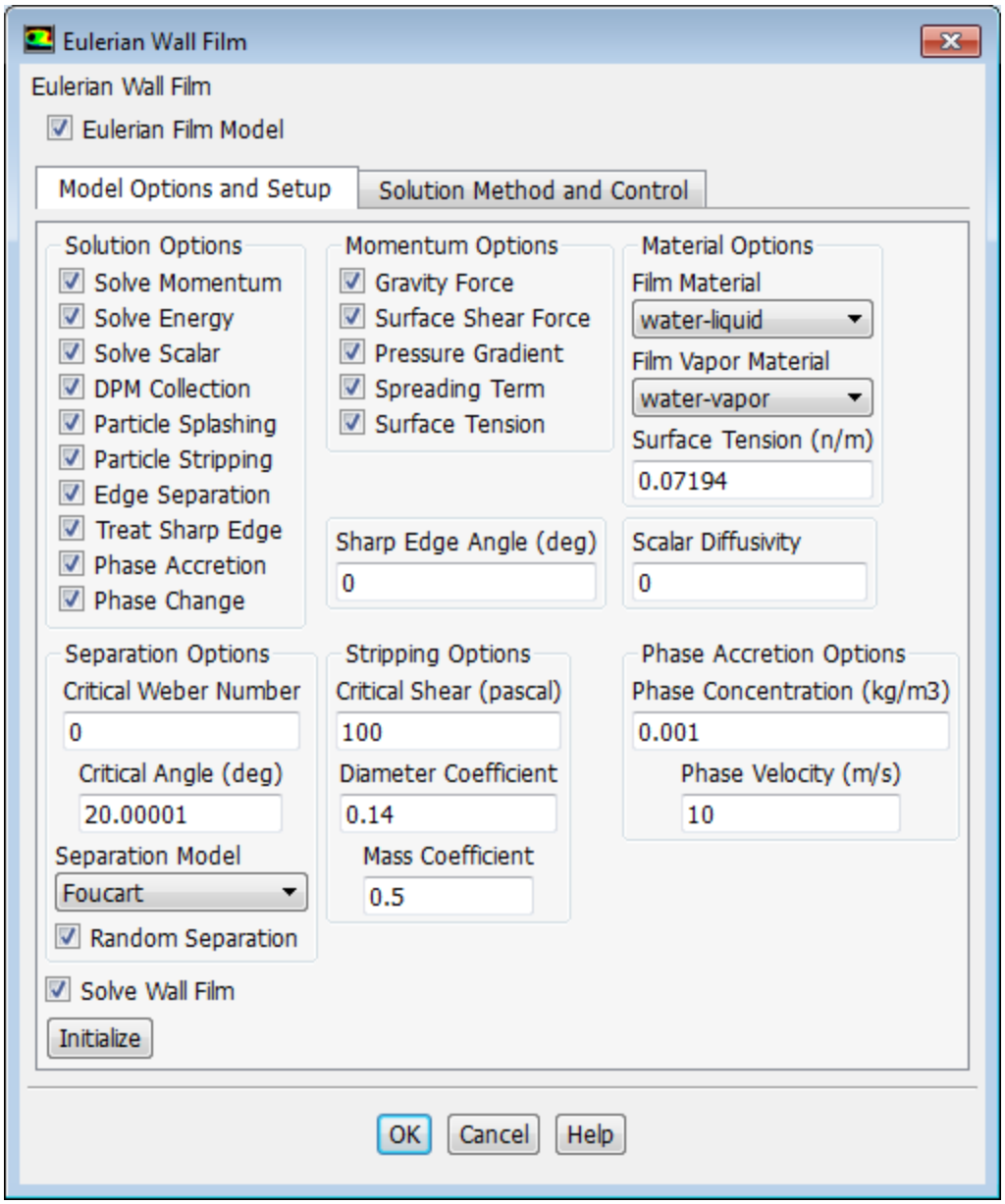
Once you open the Eulerian Wall Film Dialog Box (p. 2294), you can select the **Eulerian Wall Film** check box to enable the model so that you can use it in your simulation. Enabling the model expands the dialog box to reveal additional model options and solution controls.

Important

If you want to modify your mesh in Fluent, you must first disable the Eulerian Wall Film model. Once you save the modified mesh, you must re-enable the Eulerian Wall Film model and re-initialize your case.

You can set general Eulerian Wall Film model options in the **Model Options and Setup** tab of the **Eulerian Wall Film** dialog box. This tab contains controls for specific solution, discrete phase model (DPM), and material options for the Eulerian Wall Film model.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the **Model Options and Setup** tab, the following options and controls are available:

Solve Momentum

specifies whether the momentum equation (see Equation 19.2 in the *Fluent Theory Guide*) is solved for the wall film or not. In the **Momentum Options** group box, you can individually select each term of the equation for inclusion in the calculations.

Note that **Spreading Term** and **Surface Tension** under **Momentum Options** are available only when **Pressure Gradient** is selected.

Solve Energy

specifies whether the energy equation (see Equation 19.3 in the *Fluent Theory Guide*) is solved for the wall film or not.

Solve Scalar

specifies whether the passive scalar transport equation Equation 19.36 is solved for the wall film or not. When the **Solve Scalar** option is enabled, you can specify **Scalar Diffusivity**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DPM Collection
includes discrete phase particles and their interaction with the film model (see DPM Collection in the *Fluent Theory Guide*). You can select:

- **Particle Splashing**

  You can specify splashing options in the **Wall Film** tab of the **Wall** dialog box. For details, see Setting Eulerian Wall Film Boundary, Initial, and Source Term Conditions (p. 1542).

- **Particle Stripping** and specify the settings under the **Stripping Options** group box (for details, see Film Separation in the *Fluent Theory Guide*)

- **Edge Separation** and specify the settings under the **Separation Options** group box (for details, see Film Separation in the *Fluent Theory Guide*)

  Note that **Random Separation** means that the locations at which the newly spawned particles are injected (due to film separation) are randomly selected along the edge at which the separation takes place.

Treat Sharp Edge
accounts for the effects of sharp edges and specify **Sharp Edge Angle**. When the **Treat Sharp Edge** option is enabled and where the edge angle is smaller than the **Sharp Edge Angle**, the film wall edge is treated as a boundary edge (namely, the film becomes detached from the wall, rather than bending around the edge and attaching to the wall). If the Discrete Phase Model has been enabled along with the Eulerian Wall Film model, then the film wall edge is treated as a separation edge.

Phase Accretion
accounts for the effect of the interaction of the wall film with Eulerian and Mixture multiphase flow (see Secondary Phase Accretion in the *Fluent Theory Guide*). This option enables you to compute the secondary phase collection efficiency on a wall surface. This option is only available when the Eulerian or Mixture (with Slip Velocity) Multiphase model is enabled.

Note

The following considerations should be taken into account when the Eulerian Wall Film model is used with the Eulerian or Mixture multiphase models:

- You must create temporary placeholder injections in order to model edge separation and edge stripping.
- The material properties for the secondary phase should be the same as the DPM particles.
- The name of the secondary phase material should be included in the name of the DPM particle's material and should be appended with the string '-particle' (for example, if the secondary phase material name is *waterliquid- eulerian*, then the DPM particle's material name should be *waterliquid-eulerian-particle*).

Phase Change
accounts for phase changes between the film material (liquid) and the gas species (vapor) (see Coupling of Wall Film with Mixture Species Transport in the *Fluent Theory Guide*). This option enables you to set the following:

- **Film Vapor Material**—in the **Material Options** group box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Phase change parameters—in the **Wall Film** tab of the **Wall** dialog box. For details, see Setting Eulerian Wall Film Boundary, Initial, and Source Term Conditions (p. 1542).

Note

The liquid film can only be a single-component fluid. The secondary phase that is intended for the film material cannot be a mixture of species.

Material Options
allow to select film material and film vapour material (available only for Phase Change calculations) and specify wall film surface tension.

Note

The type of material for a wall film must be **Fluid**, **Droplet Particle**, or **Inert Particle** (fluid).

You can define density, viscosity, specific heat, and thermal conductivity of a wall film material using one of the following methods:
- constant
- polynomial (including piecewise linear and piecewise polynomial)
- compressible liquid (for fluid density)
- user-defined (not available for fluid specific heat)

Solve Wall Film
enables you to skip the wall film solution during the gas phase solution, but keep the variables and setup active.

Note

The wall film cannot be solved without first initializing the wall film model (using the **Initialize** button) to initialize the wall film variables and prepare the solver for the solution procedure.

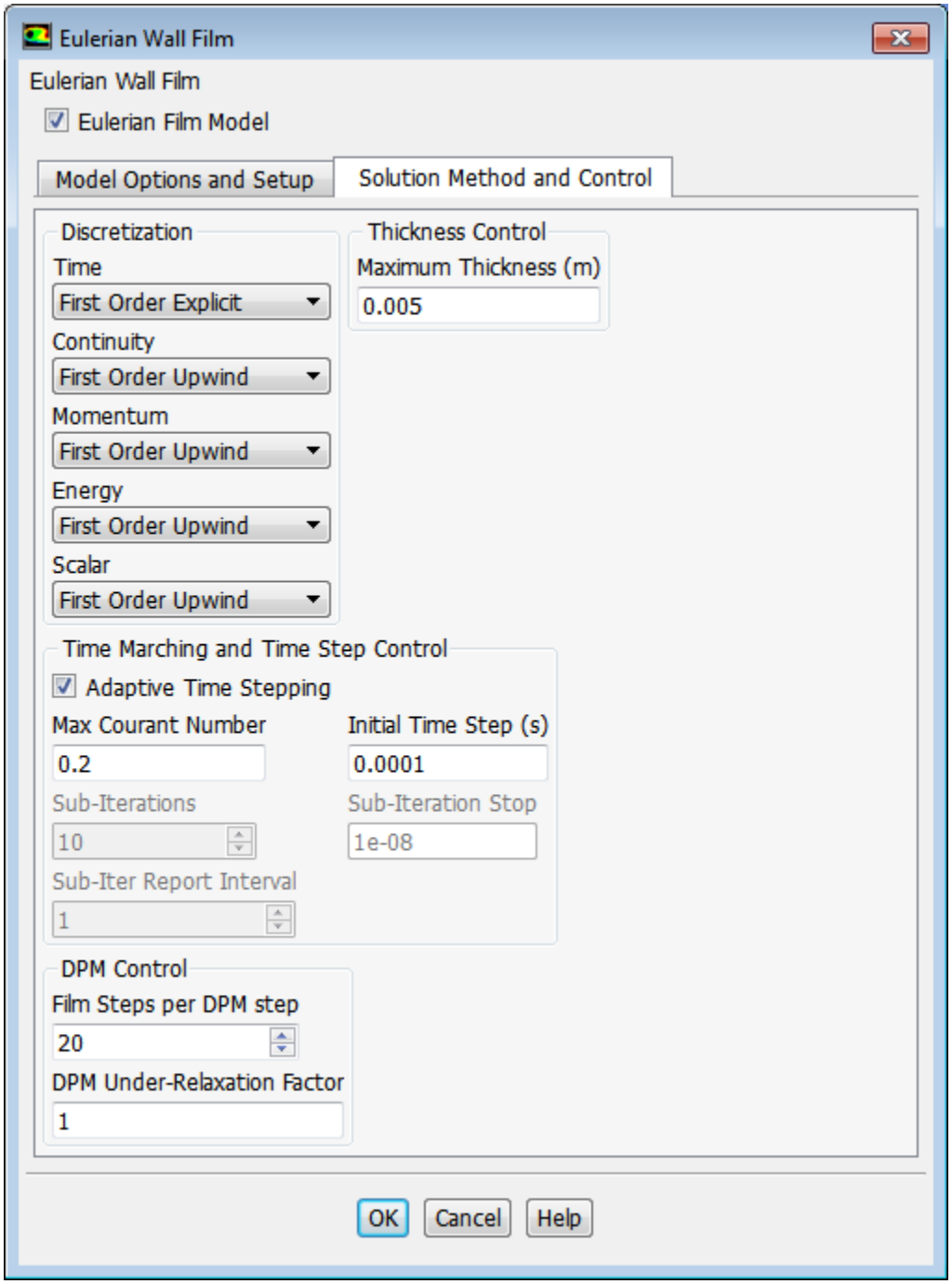
You can also use the `define/models/eulerian-wallfilm/model-options` text command to define the settings and the discretization methods for the Eulerian Wall Film model.

26.3. Setting Eulerian Wall Film Solution Controls

You can set solution controls for the Eulerian Wall Film model in the **Solution Method and Control** tab of the **Eulerian Wall Film** dialog box. This tab contains controls for specific temporal and spatial discretization options for the Eulerian Wall Film model.

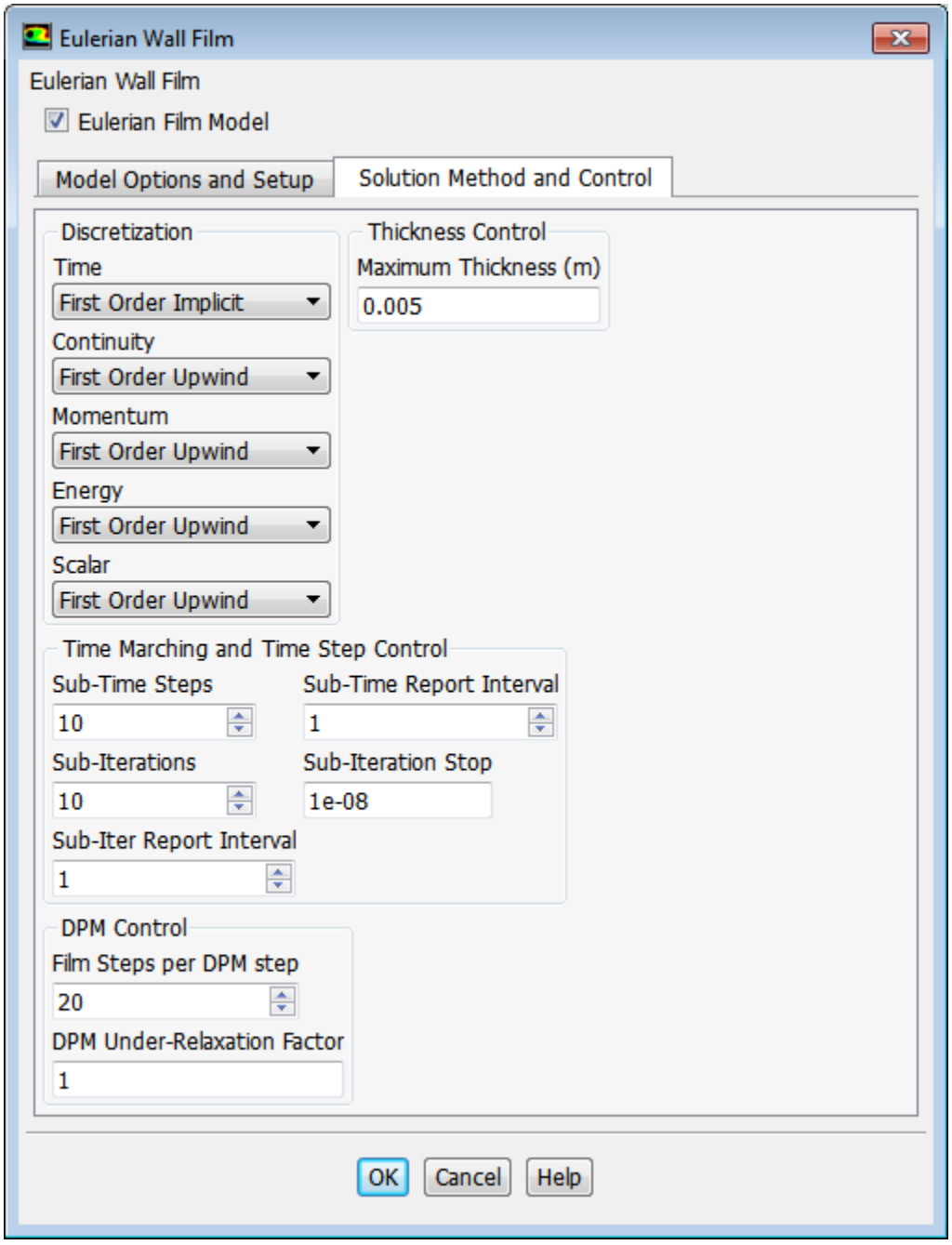
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.1: Eulerian Wall Film Solution Controls (Steady Flow)



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.2: Eulerian Wall Film Solution Controls (Unsteady Flow)



In the **Solution Method and Control** tab, you can set the following:

- Temporal and spatial (continuity, momentum, energy, and passive scalar) discretization methods under **Discretization**.
- **Maximum Thickness** for the film.

  The **Maximum Thickness** setting will limit the film thickness by removing material from the film where this value is exceeded.
- For steady state calculations (Figure 26.1: Eulerian Wall Film Solution Controls (Steady Flow) (p. 1540)):

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can specify the film model specific **Time Step** (the Eulerian Wall Film model is always transient), or you can select the **Adaptive Time Stepping** option to set the **Max. Courant Number** and the **Initial Time Step** (see Steady Flow).

- For transient cases (Figure 26.2: Eulerian Wall Film Solution Controls (Unsteady Flow) (p. 1541)):

  You can define the **Number of Time Steps** per main flow time step and the **Sub-Time Report Interval** for film calculation reporting. Note that the first and the last sub-time steps are always reported.

- For either first or second order implicit time discretization calculations:

  You can control the number of film **Sub-Iterations**, the value of **Sub-Iteration Stop** below which the film sub-iteration stops, and the **Sub-Iter Report Interval** for film sub-iteration reporting. Note that the first and the last sub-iterations are always reported.

- If **DPM Collections** is selected in the **Model Options and Setup** tab:

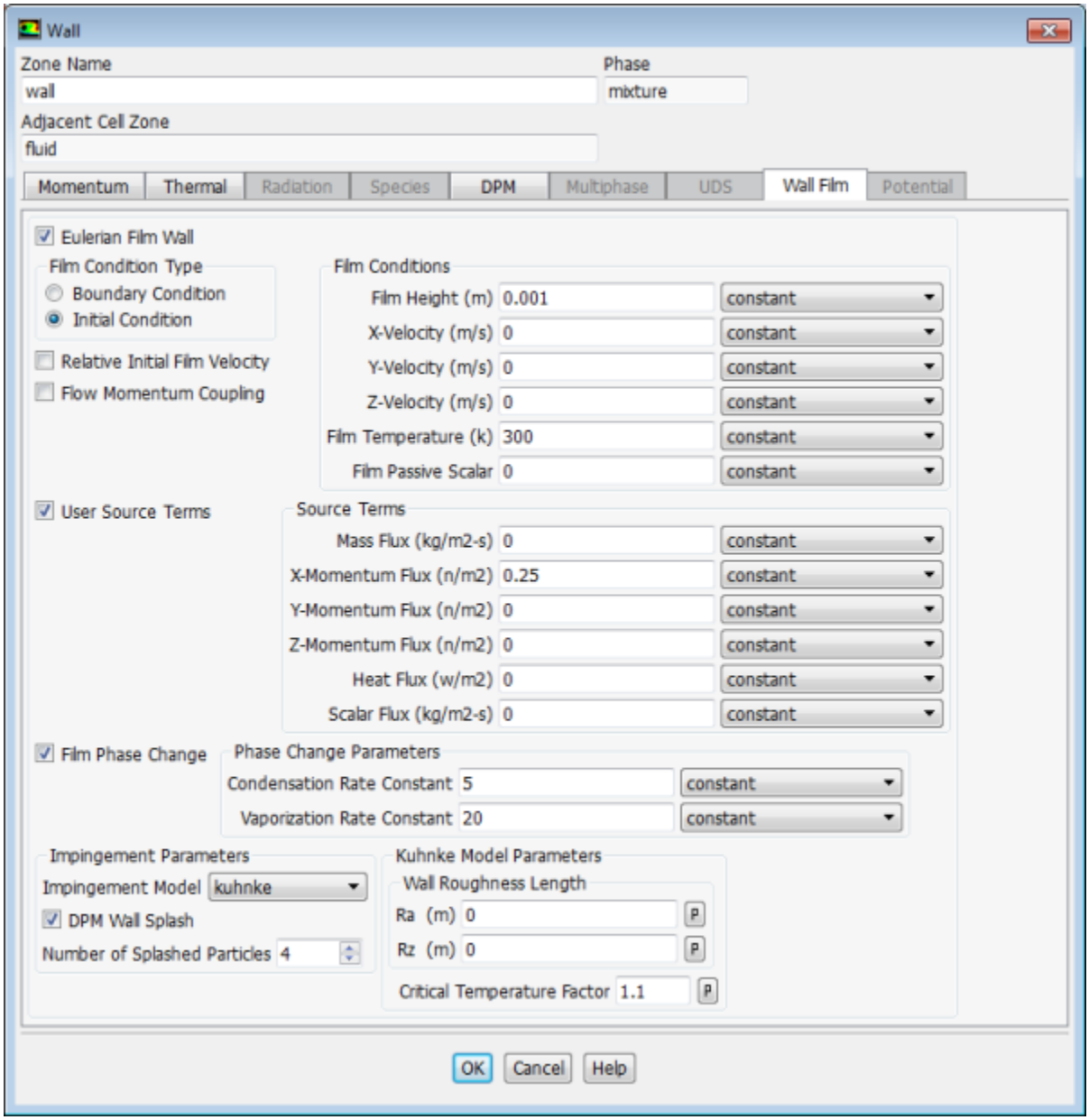
  You can set how often the DPM phase is calculated for the film by specifying a value for the **Film Steps per DPM Step** These controls are only available if **Interaction with Continuous Phase** is disabled in the **Discrete Phase Model** dialog box.

26.4. Setting Eulerian Wall Film Boundary, Initial, and Source Term Conditions

In the **Wall** dialog box, under the **Wall Film** tab, you can set boundary, initial, and source term conditions for liquid films on specified wall boundaries. When you select the **Eulerian Film Wall** option, Fluent solver designates the wall as a film wall. Wall film equations are only solved on designated film walls.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.3: Wall Dialog Box (Initial Conditions)



The following types of film conditions exist in ANSYS Fluent:

- **Boundary Condition**: Is where film mass, momentum, energy and scalar quantities are introduced to the wall film solution domain.

  You can enter values for the following quantities:

  – **Film Mass Flux**

  – **X-Momentum Flux**, **Y-Momentum Flux**, and **Z-Momentum Flux**

  – **Incoming Film Temperature**

  – **Film Passive Scalar Flux**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For moving wall simulations, you can specify the film momentum flux relative to the wall on which liquid film is defined by selecting **Relative Film Momentum Flux**. Note that this option is only available when **Moving Wall** is selected in the **Wall** dialog box and/or **Frame Motion** is selected in the **Fluid** dialog box.

the film momentum flux or initial velocity inputs will be relative to the wall on which liquid film is defined.

- **Initial Condition**: Is the initial state of wall film.

  You can set values for the following quantities:

  – **Film Height**

  – **X-Velocity**, **Y-Velocity**, and **Z-Velocity** components

  – **Film Temperature**

  – **Film Passive Scalar**

  For moving wall simulations, you can specify the initial velocity relative to the wall on which liquid film is defined by selecting **Relative Initial Film Velocity**. Note that this option is only available when **Moving Wall** is selected in the **Wall** dialog box and/or **Frame Motion** is selected in the **Fluid** dialog box.

  If you select the **User Source Terms** option (available only with initial conditions), you can also specify additional source terms to the film continuity, momentum, energy, and passive scalar equations by setting values for the following quantities:

  – **Mass Flux**

  – **X-Momentum Flux**, **Y-Momentum Flux**, and **Z-Momentum Flux**

  – **Heat Flux**

  – **Scalar Flux**

For both boundary and initial conditions, if you select the **Flow Momentum Coupling** option, the liquid film and the gas flow shares the same velocity at the interface of the liquid-gas interface using a two-way coupling. If you clear this option, the coupling between the liquid film and the gas flow occurs only one-way, namely, while the gas flow impacts the film flow, the film flow does not impact the bulk of the gas flow.

When the **Phase Change** option is selected in the **Eulerian Wall Film** dialog box, you can enable the **Film Phase Change** for the selected film wall and specify **Condensation Rate Constant** and **Vaporization Rate Constant** (C_{con} and C_{vap} in Equation 19.28 in the *Fluent Theory Guide*). You can also specify condensation and vaporization rates using user-defined profile UDFs. Note that such UDFs must return the relevant rates in kg/s. For more information on profile user-defined functions, see the separate Fluent Customization Manual.

When the **DPM Collection** option is selected in the **Eulerian Wall Film** dialog box, you can select a particle-wall impingement model and specify relevant model parameters in the **Wall** dialog box. The following impingement models are available in ANSYS Fluent:

- `stanton-rutland` (see The Stanton-Rutland Model in the *Fluent Theory Guide* for details)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- kuhnke (see The Kuhnke Model in the *Fluent Theory Guide* for details)

  For the Kuhnke model, you must specify the following model parameters:

  – **Ra**: The wall mean roughness that is used to compute K_{cold} in Equation 16.258 in the *Fluent Theory Guide*

  – **Rz**: The average surface roughness that is used to compute splashed particle reflection angle distribution Equation 16.270 in the *Fluent Theory Guide*.

  – **Critical Temperature Factor**: The dimensionless variable that is used to determine the impingement regime transition temperature in Equation 16.250 in the *Fluent Theory Guide*.

Note

You can also add an impingement effect via UDFs as described in Hooking `DEFINE_IMPINGEMENT` UDFs, Hooking `DEFINE_FILM_REGIME` UDFs, and Hooking `DEFINE_SPLASHING_DISTRIBUTION` UDFs in the *Fluent Customization Manual*. Note that UDFs settings will override those for the impingement/splashing model selected for the Eulerian wall film boundary condition.

In addition, when **Particle Splashing** is selected in the **Eulerian Wall Film** dialog box, you can select **DPM Wall Splash** and specify **Number of Splashed Particles**.

26.5. Postprocessing the Eulerian Wall Film

When using the Eulerian Wall Film model, the following additional variables will be available for postprocessing (see Field Function Definitions (p. 1931) for their definitions):

- **Film Thickness**
- **Film Mass**
- **Film Temperature** (when **Solve Energy** is enabled)
- **Film X-Velocity**
- **Film Y-Velocity**
- **Film Z-Velocity**
- **Film Velocity Magnitude**
- **Film Surface X-Velocity**
- **Film Surface Y-Velocity**
- **Film Surface Z-Velocity**
- **Film Surface Velocity Magnitude**
- **Film Surface Temperature** (when **Solve Energy** is enabled)
- **Film Passive Scalar** (when **Solve Scalar** is enabled)
- **Film Density** (when film material density is non-constant)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Film Effective Pressure**
- **Film Courant Number**
- **Film Weber Number**
- **Film Stripped Mass** (when **Particle Stripping** is enabled)
- **Film Stripped Diam** (when **Particle Stripping** is enabled)
- **Film DPM Mass Source** (when **DPM Collection** is enabled)
- **Film DPM Energy Source** (when **DPM Collection** and **Solve Energy** are enabled)
- **Film DPM X-Momentum Source** (when **DPM Collection** is enabled)
- **Film DPM Y-Momentum Source** (when **DPM Collection** is enabled)
- **Film DPM Z-Momentum Source** (when **DPM Collection** is enabled)
- **Film Separated Mass** (when **Edge Separation** is enabled)
- **Film Secondary Phase Mass** (when **Phase Accretion** is enabled)
- **Film Secondary Phase Collection Coef** (when **Phase Accretion** is enabled)

You can report on the mass and energy fluxes for the Eulerian Wall Film model using the **Flux Reports** dialog box (see Generating a Flux Report (p. 1911)), for domain boundaries such as inlets, outlets, and so on, as well as wall boundaries.

Results → Reports → Fluxes **Edit...**

The **Film Mass Flow Rate** and **Film Heat Transfer Rate** are available as options in the **Flux Reports** dialog box when the **Eulerian Wall Film** model is enabled.

Note

The energy flux report only accounts for the heat transfer rate through the external boundary. It does not consider heat transfer at gas-liquid interfaces or heat transfer due to phase change or film-DPM interaction.

You can also report on the mass flow rate and heat transfer rate using the User Interface (TUI) as follows:

- For mass flow rate, enter the text command `report/fluxes/film-mass-flow`.
- For heat transfer rate, enter the text command `report/fluxes/film-heat-transfer`.

You will be prompted for the boundary face zone IDs for which the report will be generated, and whether or not you want to save the reported values to a file. ANSYS Fluent will either print the report in the console or write it to the file you have specified.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 27: Modeling Electric Potential Field

This chapter provides details about how to model and simulate the electric potential field in ANSYS Fluent. The theoretical aspects of the electric potential model are presented in Electric Potential in the *Fluent Theory Guide*. Additional information about the model is provided in the following sections:

27.1. Overview and Limitations
27.2. Using the Electric Potential Model
27.3. Postprocessing the Electric Potential Field

27.1. Overview and Limitations

The ANSYS Fluent electric potential model allows for predicting the electric potential field and its effects. The electric potential model can be used separately (for example, to compute electric static force for charged particles in a DPM calculation) or in combination with other ANSYS Fluent models.

The electric potential solver has been integrated with the electrochemical reaction model (Electrochemical Reactions in the *Fluent Theory Guide*). When you enable the electrochemistry in your case, ANSYS Fluent automatically enables the electric potential solver. However, if you want to use the electric potential model along with other ANSYS Fluent models, you need to manually enable it as described in Using the Electric Potential Model (p. 1547).

Using the electric potential solver, you can simulate the electric potential field in both fluid and solid zones.

When defining the boundary conditions for the electric potential field, you can specify either potential or current density at the external walls.

The following limitations exist for the electric potential model:

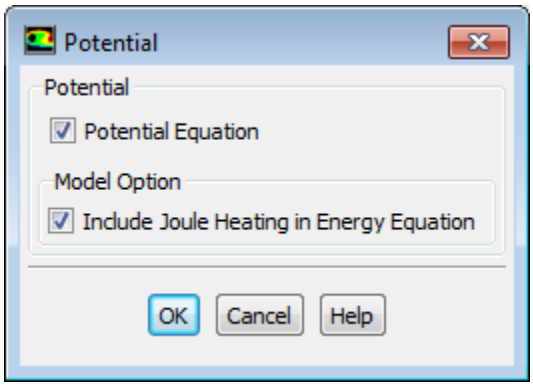
- The electric potential model cannot be used with multiphase flows.
- The electric potential model cannot be used in simulations with walls that have a shell conduction boundary condition.

27.2. Using the Electric Potential Model

The procedure for setting up the electric potential model is described below. Note that only steps that are pertinent to electric potential field modeling are listed here. You will also need to define the other settings (such as other material properties, boundary conditions, other models) as usual. For information about inputs related to other ANSYS Fluent models that you are using in conjunction with the electric potential model, refer to the appropriate sections for those models.

1. In the **Potential** dialog box, which is opened from the **Model**/**Electric Potential** tree item, enable the **Potential Equation** option.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Once the **Potential Equation** is enabled, the electric conductivity material property for fluid, solid, and mixture materials defined in your case will appear in the **Create/Edit Materials** dialog box.

2. If you want to include Joule heating in the energy equation (Equation 5.1 in the *Fluent Theory Guide*), under the **Model Option** group box, enable **Include Joule Heating in Energy Equation**.

Note

The **Include Joule Heating in Energy Equation** option appears in the **Potential** dialog box only when **Energy Equation** is enabled in the **Energy** dialog box.

---

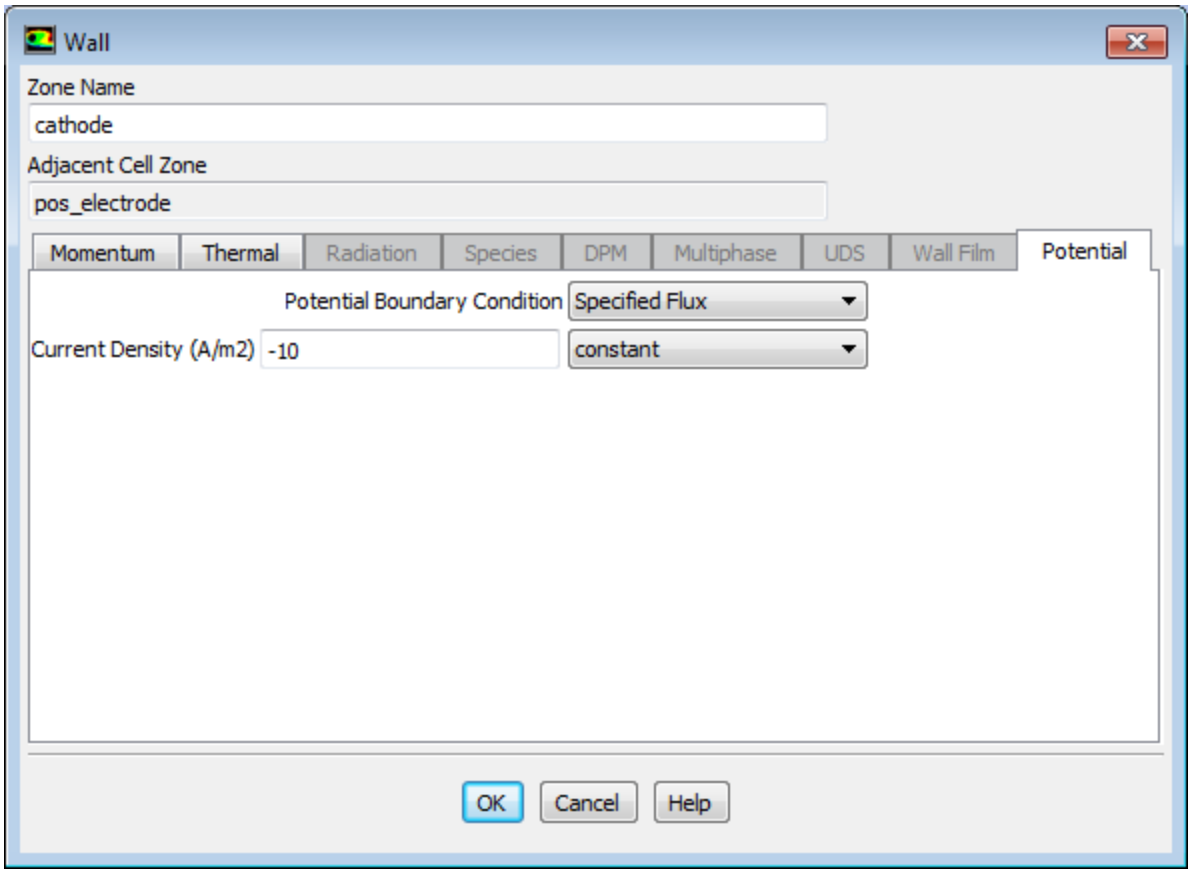
3. In the **Create/Edit Materials** dialog box, under **Properties**, specify electric conductivity for the conductive materials defined in your simulation.

Setup → **Materials**

4. Define boundary conditions on conductive external (one-sided) walls.

Setup → **Boundary Conditions**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the **Potential Boundary Condition** drop-down list, if you select:

- **Specified Flux**, you can specify the **Current Density** at the boundary wall.
- **Specified Value**, you can specify the **Potential** at the boundary wall.

5. If required, specify electric potential conditions on other boundaries in the **Potential** tab of the boundary conditions dialog boxes. Note that the default setting, zero current flux, is suitable for most problems and needs not be changed.

27.3. Postprocessing the Electric Potential Field

For potential field calculations, you can generate graphical plots or alphanumeric reports of the following items (see Field Function Definitions (p. 1931) for their definitions). These quantities are available in the variable selection drop-down lists that appear in postprocessing dialog boxes:

- **Electric Potential** (in the **Potential...** category)
- **Electric Conductivity** (in the **Properties...** category)
- **Current Magnitude** (in the **Potential...** category)
- **Joule Heat Source** (in the **Potential...** category)

You can also generate vector plots of the electric current fields by selecting **Electric Current** in the **Vectors of** drop-down list inVectors Dialog Box (p. 2831).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 28: Using the Solver

This chapter describes how to use the ANSYS Fluent solver. For more information about the theory behind the ANSYS Fluent solver, see Solver Theory in the Theory Guide. Choosing the Solver (p. 1553) provides an overview, and the remaining sections provide detailed instructions.

28.1. Overview of Using the Solver
28.2. Choosing the Spatial Discretization Scheme
28.3. Pressure-Based Solver Settings
28.4. Density-Based Solver Settings
28.5. Setting Algebraic Multigrid Parameters
28.6. Setting Solution Limits
28.7. Setting Multi-Stage Time-Stepping Parameters
28.8. Selecting Gradient Limiters
28.9. Initializing the Solution
28.10. Full Multigrid (FMG) Initialization
28.11. Hybrid Initialization
28.12. Performing Steady-State Calculations
28.13. Performing Pseudo Transient Calculations
28.14. Performing Time-Dependent Calculations
28.15. Monitoring Solution Convergence
28.16. Convergence Conditions
28.17. Executing Commands During the Calculation
28.18. Automatic Initialization of the Solution and Case Modification
28.19. Animating the Solution
28.20. Checking Your Case Setup
28.21. Convergence and Stability
28.22. Solution Steering

28.1. Overview of Using the Solver

In ANSYS Fluent, two solver technologies are available:

- pressure-based
- density-based

Both solvers can be used for a broad range of flows, but in some cases one formulation may perform better (that is, yield a solution more quickly or resolve certain flow features better) than the other. The pressure-based and density-based approaches differ in the way that the continuity, momentum, and (where appropriate) energy and species equations are solved, as described in Overview of Flow Solvers in the Theory Guide.

The pressure-based solver traditionally has been used for incompressible and mildly compressible flows. The density-based approach, on the other hand, was originally designed for high-speed compressible flows. Both approaches are now applicable to a broad range of flows (from incompressible to highly compressible), but the origins of the density-based formulation may give it an accuracy (that is shock resolution) advantage over the pressure-based solver for high-speed compressible flows.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Two formulations exist under the density-based solver: implicit and explicit. The density-based explicit and implicit formulations solve the equations for additional scalars (for example, turbulence or radiation quantities) sequentially. The implicit and explicit density-based formulations differ in the way that they linearize the coupled equations. For more details about the solver formulations, see Overview of Flow Solvers in the Theory Guide.

Due to broader stability characteristics of the implicit formulation, a converged steady-state solution can be obtained much faster using the implicit formulation rather than the explicit formulation. However, the implicit formulation requires more memory than the explicit formulation.

Two algorithms also exist under the pressure-based solver in ANSYS Fluent: a segregated algorithm and a coupled algorithm. In the segregated algorithm the governing equations are solved sequentially, segregated from one another, while in the coupled algorithm the momentum equations and the pressure-based continuity equation are solved in a coupled manner. In general, the coupled algorithm significantly improves the convergence speed over the segregated algorithm, however, the memory requirement for the coupled algorithm is more than the segregated algorithm.

When selecting a solver and an algorithm you must consider the following issues:

- The model availability for a given solver.
- Solver performance for the given flow conditions.
- The size of the mesh under consideration and the available memory on your machine. This issue could be an important factor in deciding whether to use an explicit or implicit formulation when the density-based solver is selected, or to use a segregated or coupled algorithm when the pressure-based solver is selected.

The following two lists highlight the model availability for each solver:

Important

Note that the pressure-based solver provides several physical models or features that are not available with the density-based solver:

- Cavitation model
- Volume-of-fluid (VOF) model
- Multiphase mixture model
- Eulerian multiphase model
- Non-premixed combustion model
- Premixed combustion model
- Partially premixed combustion model
- Composition PDF transport model
- Soot model
- Rosseland radiation model
- Melting/solidification model

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Shell conduction model
- Floating operating pressure
- Fixed variable option
- Physical velocity formulation for porous media
- Specified mass flow rate for streamwise periodic flow

The following features are available with the density-based solver, but not with the pressure-based solver:

- Wet steam multiphase model

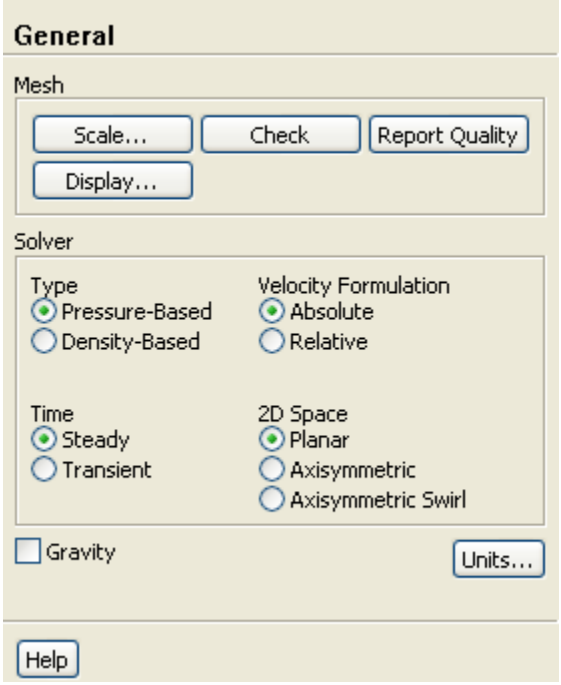
For additional information, see the following sections:
28.1.1. Choosing the Solver

28.1.1. Choosing the Solver

To choose one of the solvers, you will use the General Task Page (p. 2160) (Figure 28.1: The General Task Page (p. 1553)).

Setup → General

Figure 28.1: The General Task Page



To use the pressure-based solver, retain the default selection of **Pressure-Based** under **Solver**.

To use the density-based solver, select **Density-Based** under **Solver**.

After you have defined your model and specified which solver you want to use, you are ready to run the solver. The following steps outline a general procedure you can follow:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. (pressure-based solver only) Select the pressure-velocity coupling method (see Choosing the Pressure-Velocity Coupling Method (p. 1561)).

2. Choose the spatial discretization scheme and, for the pressure-based solver, the pressure interpolation scheme (see Choosing the Spatial Discretization Scheme (p. 1554)).

3. (pressure-based solver only) Select the porous media velocity method (see Porous Media Conditions (p. 261)).

4. Select how you want the derivatives to be evaluated by choosing a gradient option (see Evaluation of Gradients and Derivatives in the Theory Guide).

5. Set the under-relaxation factors (see Setting Under-Relaxation Factors (p. 1564)).

6. (density-based explicit formulation only) Set up the FAS multigrid (see Turning On FAS Multigrid (p. 1575)).

7. Make any additional modifications to the solver settings that are suggested in the chapters or sections that describe the models you are using.

8. Enable the appropriate solution monitors (see Monitoring Solution Convergence (p. 1624)).

9. Initialize the solution (see Initializing the Solution (p. 1591)).

10. Start calculating (see Performing Steady-State Calculations (p. 1600) for steady-state calculations, or Performing Time-Dependent Calculations (p. 1608) for time-dependent calculations).

11. If you have convergence trouble, try one of the methods discussed in Convergence and Stability (p. 1667).

The default settings for the first three items listed above are suitable for most problems and need not be changed. The following sections outline how these and other solution parameters can be changed, and when you may want to change them.

28.2. Choosing the Spatial Discretization Scheme

Gradients are needed not only for constructing values of a scalar at the cell faces, but also for computing secondary diffusion terms and velocity derivatives. For more information about the different gradients, see Evaluation of Gradients and Derivatives in the Theory Guide.

The three gradients that are available in ANSYS Fluent are

- **Green-Gauss Cell Based**
- **Green-Gauss Node Based**
- **Least Squares Cell Based**

The gradient options are selectable from the **Gradient** drop-down list, in the **Solution Methods** task page.

Solution → **Methods**

In addition, ANSYS Fluent allows you to choose the discretization scheme for the convection terms of each governing equation. (Second-order accuracy is automatically used for the viscous terms.) By default, single-phase problems using either the pressure-based or density-based solver are solved using second-order upwind discretization for the convection terms of the flow equations and all scalar equations except those for turbulence quantities, which are solved using first-order upwind discretization. For

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

multiphase flows, the flow equations use first-order upwind discretization by default. For a complete description of the discretization schemes available in ANSYS Fluent, see Discretization in the Theory Guide.

In addition, when you use the pressure-based solver, you can specify the pressure interpolation scheme. For a description of the pressure interpolation schemes available in ANSYS Fluent, see Pressure Interpolation Schemes in the Theory Guide.

For additional information, see the following sections:

28.2.1. First-Order Accuracy vs. Second-Order Accuracy
28.2.2. Other Discretization Schemes
28.2.3. Choosing the Pressure Interpolation Scheme
28.2.4. Choosing the Density Interpolation Scheme
28.2.5. High Order Term Relaxation (HOTR)
28.2.6. User Inputs

28.2.1. First-Order Accuracy vs. Second-Order Accuracy

When the flow is aligned with the mesh (for example, laminar flow in a rectangular duct modeled with a quadrilateral or hexahedral mesh) the first-order upwind discretization may be acceptable. When the flow is not aligned with the mesh (that is, when it crosses the mesh lines obliquely), however, first-order convective discretization increases the numerical discretization error (numerical diffusion). For triangular and tetrahedral meshes, since the flow is never aligned with the mesh, you will generally obtain more accurate results by using the second-order discretization. For quad/hex meshes, you will also obtain better results using the second-order discretization, especially for complex flows.

In summary, while the first-order discretization generally yields better convergence than the second-order scheme, it generally will yield less accurate results, especially on tri/tet meshes. See Convergence and Stability (p. 1667) for information about controlling convergence.

For most cases, you will be able to use the second-order scheme from the start of the calculation. In some cases, however, you may need to start with the first-order scheme and then switch to the second-order scheme after a few iterations. For example, if you are running a high-Mach-number flow calculation that has an initial solution much different than the expected final solution, you will usually need to perform a few iterations with the first-order scheme and then turn on the second-order scheme and continue the calculation to convergence. Alternatively, full multigrid initialization is also available for some flow cases that allow you to proceed with the second-order scheme from the start.

For a simple flow that is aligned with the mesh (for example, laminar flow in a rectangular duct modeled with a quadrilateral or hexahedral mesh), the numerical diffusion will be naturally low, so you can generally use the first-order scheme instead of the second-order scheme without any significant loss of accuracy.

Finally, if you run into convergence difficulties with the second-order scheme, you should try the first-order scheme instead.

28.2.1.1. First-to-Higher Order Blending

While the higher-order scheme may result in greater accuracy, it can also result in convergence difficulties and instabilities at certain flow conditions. On the other hand, using a first-order scheme may not provide the desired accuracy. One approach to achieving improved accuracy while maintaining good stability is to use a discretization blending factor. This feature is available for both density-based and pressure-based solvers and can be invoked using the following text command:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`solve → set → numerics`

Enter a value between `0` and `1` when asked for the blending factor: `1st-order to higher-order blending factor [min=0.0 - max=1.0]`

A blending factor of `0` reduces the gradient reconstruction to a first-order discretization scheme, whereas `1` will recover high-order discretization. A blending factor of less than `1` (typically 0.75 or 0.5) will make the convective fluxes more diffusive, which in some flow conditions can stabilize a solution that is otherwise unstable when the full higher-order discretization scheme is employed.

Important

Note that in order to use this feature effectively, make sure that one of the allowed higher order discretization schemes is selected for the desired variables in the **Solution Methods** task page.

28.2.2. Other Discretization Schemes

The QUICK and third-order MUSCL discretization schemes may provide better accuracy than the second-order scheme for rotating or swirling flows. The QUICK scheme is applicable to quadrilateral or hexahedral meshes, while the MUSCL scheme is used on all types of meshes. In general, however, the second-order scheme is sufficient and the QUICK scheme will not provide significant improvements in accuracy.

Important

If QUICK is used for hybrid meshes, it will be used only for quadrilateral and hexahedral cells. Second-order upwind discretization will be applied to all other cells.

A power law scheme is also available, but it will generally yield the same accuracy as the first-order scheme.

The bounded central differencing and central differencing schemes are available only when you are using the LES, DES, SAS, SBES, and SDES turbulence models, and the central differencing scheme should be used only when the mesh spacing is fine enough so that the magnitude of the local Peclet number (see Equation 21.6 in the Theory Guide) is less than 1.

A modified HRIC scheme (see Modified HRIC Scheme in the Theory Guide) is also available for VOF simulations using either the implicit or explicit formulation.

28.2.3. Choosing the Pressure Interpolation Scheme

As discussed in Pressure Interpolation Schemes in the Theory Guide, a number of pressure interpolation schemes are available when the pressure-based solver is used in ANSYS Fluent. For most cases the second-order scheme is acceptable, but some types of models may benefit from one of the other schemes:

- For mixture or VOF multiphase models, either the PRESTO! or body-force-weighted schemes should be used and only these schemes are made available. The default in these cases is PRESTO!.
- For problems involving large body forces, the body-force-weighted scheme is recommended.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For flows with high swirl numbers, high-Rayleigh-number natural convection, high-speed rotating flows, and flows in strongly curved domains, use the PRESTO! scheme.

Note that you will not specify the pressure interpolation scheme if you are using the Eulerian multiphase model. ANSYS Fluent will use the solution method described in Solution Method in ANSYS Fluent in the Theory Guide for Eulerian multiphase calculations.

28.2.4. Choosing the Density Interpolation Scheme

As discussed in Density Interpolation Schemes in the Theory Guide, four density interpolation schemes are available when the pressure-based solver is used to solve a single-phase compressible flow.

The second-order upwind scheme (the default) provides reasonable stability for the discretization of the pressure-correction equation, and gives good results for most classes of flows. The first-order upwind scheme will provide greater stability, but may tend to smooth shocks in compressible flows. If you are calculating a compressible flow with shocks you should use the second-order-upwind or QUICK scheme. Using the QUICK scheme for all variables, including density, is highly recommended for compressible flows with shocks when using quadrilateral, hexahedral, or hybrid meshes. The third-order MUSCL scheme is applicable to arbitrary meshes and has the potential to improve spatial accuracy for all types of meshes by reducing numerical diffusion.

Important

In the case of multiphase flows, the selected density scheme is applied to the compressible phase and arithmetic averaging is used for incompressible phases.

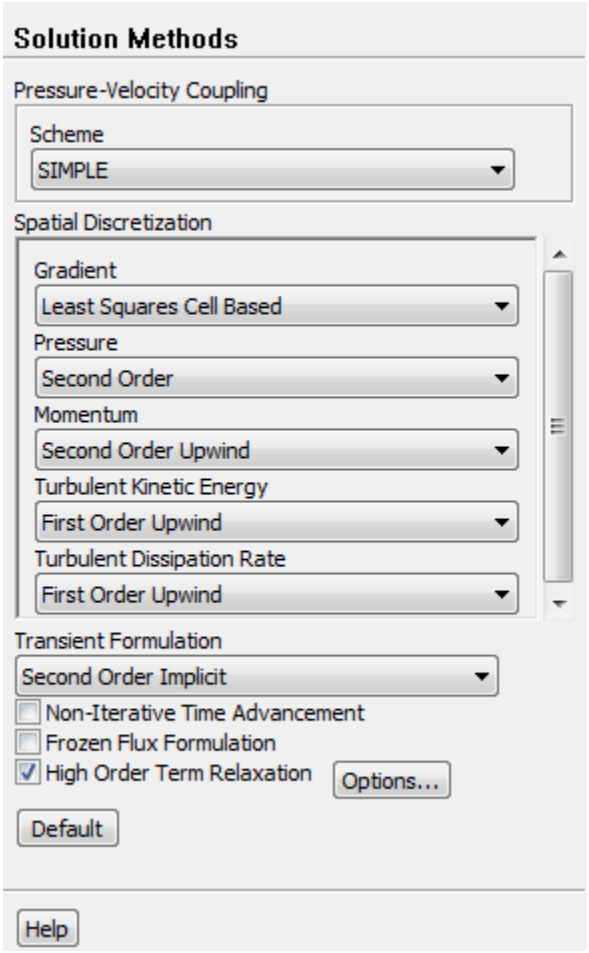
28.2.5. High Order Term Relaxation (HOTR)

The purpose of the relaxation of high order terms is to improve the startup and the general solution behavior of flow simulations when higher order spatial discretizations are used (higher than first). It has also shown to prevent convergence stalling in some cases. Such high-order terms can be of significant importance in certain cases and lead to numerical instabilities. This is particularly true at aggressive solution settings. In such cases, high order relaxation is a useful strategy to minimize your interaction during the solution. This can be an effective alternative to starting the solution first order, then switching to second order spatial discretization at a later stage.

The **High Order Term Relaxation** option can be enabled from the **Solution Methods** task page, as shown in Figure 28.2: The Solution Methods Task Page for the HOTR Option (p. 1558).

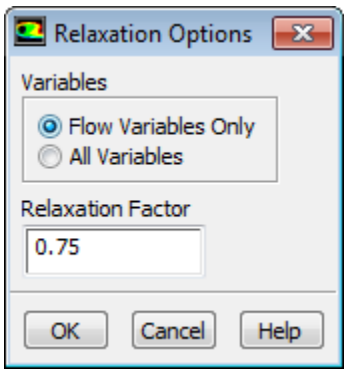
Solution → **Methods**

Figure 28.2: The Solution Methods Task Page for the HOTR Option



Further control of **High Order Term Relaxation** can be obtained after clicking **Options...** and making the necessary selections and settings in the **Relaxation Options** dialog box (Figure 28.3: The Relaxation Options Dialog Box (p. 1558)).

Figure 28.3: The Relaxation Options Dialog Box



You have the option of selecting **All Variables** to be under-relaxed instead of only the default flow variables (**Flow Variables Only**).

- If you select **Flow Variables Only**, then the following variables will be under-relaxed:
  - Velocity components

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - Pressure
  - Energy
  - Density
  - Turbulence quantities (excluding Reynolds stresses)
  - Volume fraction

- If you select **All Variables**, then relaxation is applied to each variable discretized with a higher order scheme.

The default values for the **Relaxation Factor** is 0.25 for steady-state cases and 0.75 for transient cases. The same factor is applied to all equations solved.

For theoretical information about high order term relaxation, see High Order Term Relaxation in the Theory Guide.

28.2.5.1. Limitations

The following limitations exist when using the **High Order Term Relaxation** option:

- The **High Order Term Relaxation** option is not available when the **Non-Iterative Time Advancement** option is enabled, since the simulation would not achieve the required level of high order spatial accuracy.
- In general, high order term relaxation is available for transient flows. Nevertheless, it should be used with care. To achieve high order accuracy at convergence for each time step, you must increase the number of iterations per time step to ensure that the original convergence criteria have been met.
- When the QUICK scheme is selected for specific transport equations, no under-relaxation is applied to this equation.

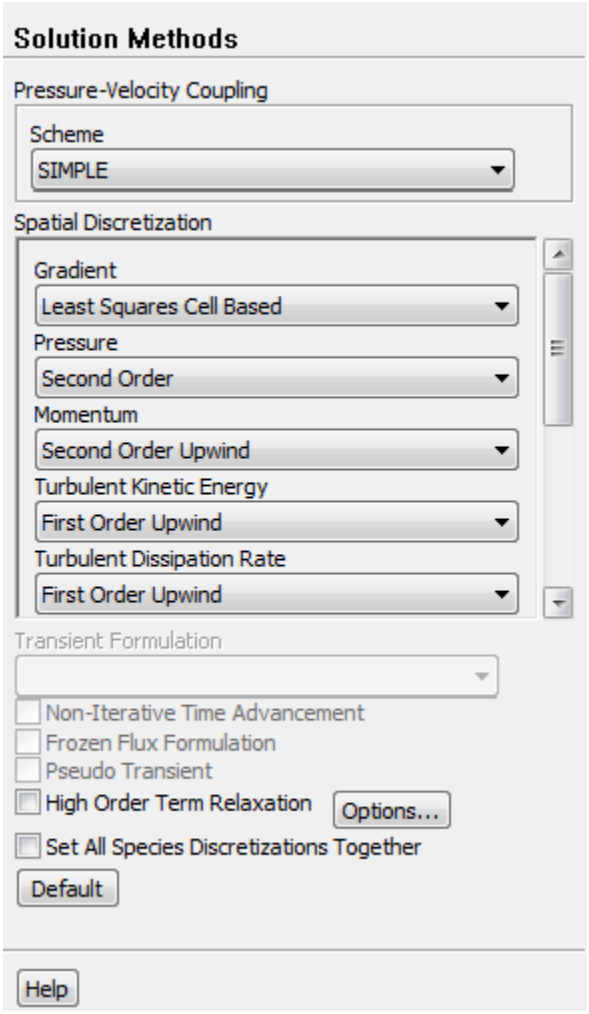
28.2.6. User Inputs

You can specify the discretization scheme and, for the pressure-based solver, the pressure interpolation scheme in the Solution Methods Task Page (p. 2504) (Figure 28.4: The Solution Methods Task Page for the Pressure-Based Segregated Algorithm (p. 1560)).

Solution → **Methods**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.4: The Solution Methods Task Page for the Pressure-Based Segregated Algorithm



For each scalar equation listed under **Spatial Discretization** (**Momentum**, **Energy**, **Turbulent Kinetic Energy**, and so on, for the pressure-based solver or **Turbulent Kinetic Energy**, **Turbulent Dissipation Rate**, and so on, for the density-based solver) you can choose **First Order Upwind**, **Second Order Upwind**, **Power Law**, **QUICK**, **Third-Order MUSCL**, or (if you are using the LES turbulence model) **Bounded Central Differencing** (the default) or **Central Differencing** in the adjacent drop-down list. For the density-based solver, you can choose either **First Order Upwind**, **Second Order Upwind**, or **Third-Order MUSCL** for the **Flow** equations (which include momentum and energy). Note that the task page shown in Figure 28.4: The Solution Methods Task Page for the Pressure-Based Segregated Algorithm (p. 1560) is for the pressure-based solver.

If you are using the pressure-based solver, select the pressure interpolation scheme under **Spatial Discretization**, in the drop-down list next to **Pressure**. You can choose **Standard**, **PRESTO!**, **Linear**, **Second Order**, or **Body Force Weighted**.

Important

The low-order formulation of PRESTO! can be applied by limiting the high-order terms for the PRESTO! scheme. This is done using the following text command:

`solve → set → numerics`

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When asked `limit high-order terms for PRESTO! pressure scheme?`, enter `yes`.

This modification can be used to stabilize the solution process when the pressure-based coupled algorithm is used and when the original PRESTO! scheme either fails to converge or produces unphysical oscillations (wiggles) in the pressure field.

If you are using the pressure-based solver and your flow is compressible (that is, you are using the ideal gas law for density), select the density interpolation scheme under **Spatial Discretization**, in the drop-down list next to **Density**. You can choose **First Order Upwind**, **Second Order Upwind**, **QUICK** or **Third-Order MUSCL**. (Note that **Density** will not appear for incompressible flows.)

If you enable the VOF model while using the pressure-based solver, the volume fraction interpolation schemes that are available are **Geo-Reconstruct**, **CICSAM**, **Modified HRIC**, and **QUICK**.

If your case involves species transport, you can set the scheme for the individual species as **First Order Upwind**, **Second Order Upwind**, **Power Law**, **QUICK**, or **Third-Order MUSCL**. However, if you want all your species to use the same discretization scheme, then rather than setting each one individually, simply enable the **Set All Species Discretizations Together** option. Notice that you will no longer see your list of individual species, instead a **Species** field will appear with the scheme of your choice.

If you change the settings for the **Spatial Discretization**, but you then want to return to ANSYS Fluent's default settings, you can click the **Default** button.

Important

If your mesh topology has step-wise prism mesh near the walls, do not use node-based gradients with MUSCL.

28.3. Pressure-Based Solver Settings

For additional information, see the following sections:

28.3.1. Choosing the Pressure-Velocity Coupling Method
28.3.2. Setting Under-Relaxation Factors
28.3.3. Setting Solution Controls for the Non-Iterative Solver

28.3.1. Choosing the Pressure-Velocity Coupling Method

ANSYS Fluent provides four segregated types of algorithms: SIMPLE, SIMPLEC, PISO, and (for time-dependent flows using the **Non-Iterative Time Advancement** option (NITA)) Fractional Step (FSM). These schemes are referred to as the pressure-based segregated algorithm. Steady-state calculations will generally use SIMPLE or SIMPLEC, while PISO is recommended for transient calculations. PISO may also be useful for steady-state and transient calculations on highly skewed meshes. In ANSYS Fluent, using the **Coupled** algorithm enables full pressure-velocity coupling, hence it is referred to as the pressure-based coupled algorithm.

Important

Pressure-velocity coupling is relevant only for the pressure-based solver.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.3.1.1. SIMPLE vs. SIMPLEC

In ANSYS Fluent, both the standard SIMPLE algorithm and the SIMPLEC (SIMPLE-Consistent) algorithm are available. SIMPLE is the default, but many problems will benefit from using SIMPLEC, particularly because of the increased under-relaxation that can be applied, as described below.

For relatively uncomplicated problems (laminar flows with no additional models enabled) in which convergence is limited by the pressure-velocity coupling, you can often obtain a converged solution more quickly using SIMPLEC. With SIMPLEC, the pressure-correction under-relaxation factor is generally set to 1.0, which aids in convergence speed-up. In some problems, however, increasing the pressure-correction under-relaxation to 1.0 can lead to instability due to high mesh skewness. For such cases, you will need to use one or more skewness correction schemes, use a slightly more conservative under-relaxation value (up to 0.7), or use the SIMPLE algorithm. For complicated flows involving turbulence and/or additional physical models, SIMPLEC will improve convergence only if it is being limited by the pressure-velocity coupling. Often it will be one of the additional modeling parameters that limits convergence; in this case, SIMPLE and SIMPLEC will give similar convergence rates.

28.3.1.2. PISO

The PISO algorithm (see PISO in the Theory Guide) with neighbor correction is highly recommended for all transient flow calculations, especially when you want to use a large time step. (For problems that use the LES turbulence model, which usually requires small time steps, using PISO may result in an increased computational expense, so SIMPLE or SIMPLEC should be considered instead.) PISO can maintain a stable calculation with a larger time step and an under-relaxation factor of 1.0 for both momentum and pressure. For steady-state problems, PISO with neighbor correction does not provide any noticeable advantage over SIMPLE or SIMPLEC with optimal under-relaxation factors.

PISO with skewness correction is recommended for both steady-state and transient calculations on meshes with a high degree of distortion.

When you use PISO neighbor correction, under-relaxation factors of 1.0 or near 1.0 are recommended for all equations. If you use just the PISO skewness correction for highly-distorted meshes (without neighbor correction), set the under-relaxation factors for momentum and pressure so that they sum to 1 (for example, 0.3 for pressure and 0.7 for momentum). If you use both PISO methods, follow the under-relaxation recommendations for PISO neighbor correction, above.

For most problems, it is not necessary to disable the default coupling between neighbor and skewness corrections. For highly distorted meshes, however, disabling the default coupling between neighbor and skewness corrections is recommended.

28.3.1.3. Fractional Step Method

The Fractional Step method (FSM), described in Fractional-Step Method (FSM) in the Theory Guide, is available when you choose to use the NITA scheme (that is, the **Non-Iterative Time Advancement** option in the **Solution Methods** task page). With the NITA scheme, the FSM is slightly less computationally expensive compared to the PISO algorithm. Whether you select FSM or PISO depends on the application. For some problems (for example, simulations that use VOF), FSM could be less stable than PISO.

In most cases, the default values for the solution methods are enough to set a robust convergence of the internal pressure correction sub-iterations due to skewness. Only very complex problems (for example, moving deforming meshes, sliding interfaces, the VOF model) could require a reduction of relaxation for pressure up to a value of 0.7 or 0.8.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.3.1.4. Coupled

Selecting **Coupled** from the **Pressure-Velocity Coupling** drop-down list indicates that you are using the pressure-based coupled algorithm, described in Coupled Algorithm in the Theory Guide. This solver offers some advantages over the pressure-based segregated algorithm. The pressure-based coupled algorithm obtains a more robust and efficient single phase implementation for steady-state flows. It is not available for cases using the **Non-Iterative Time Advancement** option (NITA).

Note

In some cases using porous jump boundary conditions, the **Coupled** scheme may suffer from convergence issues that do not respond to changes in the coupled solver settings. This behavior depends on the specific flow configuration and porous jump boundary condition values. If convergence instability is observed in cases using porous jump boundary conditions and the **Coupled** scheme, it is recommended that you change the pressure-velocity coupling to one of the segregated schemes.

28.3.1.5. User Inputs

You can specify the pressure-velocity coupling method in the Solution Methods Task Page (p. 2504) (Figure 28.4: The Solution Methods Task Page for the Pressure-Based Segregated Algorithm (p. 1560)).

Solution → **Methods**

Choose **SIMPLE**, **SIMPLEC**, **PISO**, **Fractional Step**, or **Coupled** in the **Pressure-Velocity Coupling** drop-down list.

If you choose **PISO**, the task page will expand to show the additional parameters for pressure-velocity coupling. By default, the number of iterations for **Skewness Correction** and **Neighbor Correction** are set to `1`. If you want to use only **Skewness Correction**, then set the number of iterations for **Neighbor Correction** to `0`. Likewise, if you want to use only **Neighbor Correction**, then set the number of iterations for **Skewness Correction** to `0`. For most problems, you do not need to change the default iteration values. By default, the **Skewness-Neighbor Coupling** option is enabled to allow for a more economical, but a less robust variation of the PISO algorithm.

If you choose **SIMPLEC** under **Pressure-Velocity Coupling**, you must also set the **Skewness Correction**, whose default value is `0`.

If you choose **Coupled**, you will have to specify the **Courant** number in the **Solution Controls** task page, which is set at `200` by default. You will also specify the **Explicit Relaxation Factors** for **Momentum** and **Pressure**, which are set at `0.75` by default. For more information about these options, refer to Pressure-Velocity Coupling and Steady-State Iterative Algorithm in the Theory Guide.

If high-order momentum discretization is used, you may need to decrease the explicit relaxation to `0.5`. For cases with very skewed meshes, the run can be stabilized by further reduction of the explicit relaxation factor to `0.25`. If ANSYS Fluent immediately diverges in the AMG solver, then the CFL number is too high and should be reduced. Reducing the CFL number below `10` is not recommended since it would be better to use the segregated algorithm for the pressure-velocity coupling.

In most transient cases, the CFL number should be set to 10^7 with an explicit relaxation of 1.0.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you choose **Coupled** and enable the **Pseudo Transient** option, you will set the **Pseudo Transient Explicit Relaxation Factors** in the **Solution Controls** task page, as described in Setting Pseudo Transient Explicit Relaxation Factors (p. 1602).

28.3.2. Setting Under-Relaxation Factors

The pressure-based solver uses under-relaxation of equations to control the update of computed variables at each iteration (as described in Under-Relaxation of Equations in the Theory Guide). This means that all equations solved using the pressure-based solver, *including the non-coupled equations solved by the density-based solver* (turbulence and other scalars, as discussed in Density-Based Solver in the Theory Guide), will have under-relaxation factors associated with them.

In ANSYS Fluent, the default under-relaxation parameters for all variables are set to values that are near optimal for the largest possible number of cases. These values are suitable for many problems, but for some particularly nonlinear problems (for example, some turbulent flows or high-Rayleigh-number natural-convection problems) it is prudent to reduce the under-relaxation factors initially.

It is good practice to begin a calculation using the default under-relaxation factors. If the residuals continue to increase after the first 4 or 5 iterations, you should reduce the under-relaxation factors.

Occasionally, you may make changes in the under-relaxation factors and resume your calculation, only to find that the residuals begin to increase. This often results from increasing the under-relaxation factors too much. A cautious approach is to save a data file before making any changes to the under-relaxation factors, and to give the solution algorithm a few iterations to adjust to the new parameters. Typically, an increase in the under-relaxation factors brings about a slight increase in the residuals, but these increases usually disappear as the solution progresses. If the residuals jump by a few orders of magnitude, you should consider halting the calculation and returning to the last good data file saved.

Note that viscosity and density are under-relaxed from iteration to iteration. Also, if the enthalpy equation is solved directly instead of the temperature equation (that is, for non-premixed combustion calculations), the update of temperature based on enthalpy will be under-relaxed. To see the default under-relaxation factors, you can click the **Default** button in the Solution Controls Task Page (p. 2508).

For most flows, the default under-relaxation factors do not usually require modification. If unstable or divergent behavior is observed, however, you need to reduce the under-relaxation factors for pressure, momentum, k, and ε from their default values to about 0.2, 0.5, 0.5, and 0.5. (It is usually not necessary to reduce the pressure under-relaxation for SIMPLEC.) In problems where density is strongly coupled with temperature, as in very-high-Rayleigh-number natural- or mixed-convection flows, it is wise to also under-relax the temperature equation and/or density (that is, use an under-relaxation factor less than 1.0). Conversely, when temperature is not coupled with the momentum equations (or when it is weakly coupled), as in flows with constant density, the under-relaxation factor for temperature can be set to 1.0.

For other scalar equations (for example, swirl, species, mixture fraction and variance) the default under-relaxation may be too aggressive for some problems, especially at the start of the calculation. You may want to reduce the factors to 0.8 to facilitate convergence.

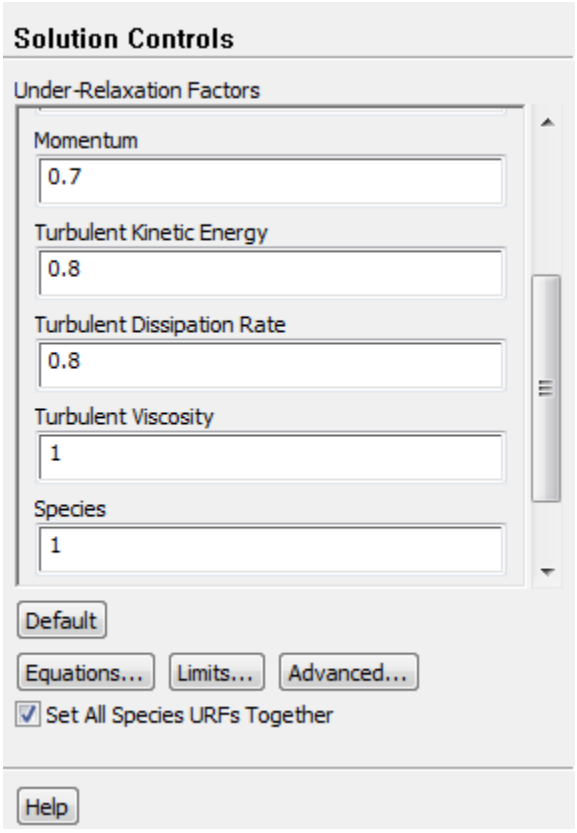
28.3.2.1. User Inputs

You can modify the under-relaxation factors in the Solution Controls Task Page (p. 2508) (Figure 28.5: The Solution Controls Task Page for the Pressure-Based Solver (p. 1565)).

Solution → **Controls**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.5: The Solution Controls Task Page for the Pressure-Based Solver



You can set the under-relaxation factor for each equation in the field next to its name under **Under-Relaxation Factors**.

Important

If you are using the pressure-based solver, all equations will have an associated under-relaxation factor (see Under-Relaxation of Equations in the Theory Guide). If you are using the density-based solver, only those equations that are solved sequentially (see Density-Based Solver in the Theory Guide) will have under-relaxation factors.

If your case involves species transport, you can set the under-relaxation factors for each of the listed species. If you want all your species to use the same under-relaxation factors, simply enable the **Set All Species URFs Together** option. Notice that you will no longer see your list of individual species, instead a **Species** field will appear where you will specify the under-relaxation factor.

If you change under-relaxation factors, but you then want to return to ANSYS Fluent's default settings, you can click the **Default** button.

Note that with optimal settings, the convergence of the coupled pressure-velocity algorithm will be limited by the segregated solution of other scalar equations, for example, turbulence. For optimum solver performance, you will need to increase the relaxation factors for these equations to a value greater than the default values.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.3.3. Setting Solution Controls for the Non-Iterative Solver

You can use the non-iterative solver (see Time-Advancement Algorithm in the Theory Guide) for transient problems in order to increase the speed and efficiency of the calculations. You can also obtain a steady solution using the non-iterative multiphase solver for multiphase flows with quasi-steady conditions.

The settings for the non-iterative solver should provide control over the maximum number of sub-iterations for each individual equation. The criteria for convergence include the **Correction Tolerance** (defined by the overall accuracy), **Residual Tolerance** (controlling the solution of the linear equations), and the individual **Relaxation Factor**. The default control settings are optimally designed in order to get a second-order accurate solution. These controls are accessible via the **Expert** tab, in the **Advanced Solution Controls** dialog box (Figure 28.6: The Advanced Solution Controls Dialog Box for the Pressure-Based Segregated Non-Iterative Solver (p. 1567)).

To use ANSYS Fluent's non-iterative transient solver in order to boost the efficiency of transient simulations:

1. Go to the **Solution Methods** task page.

   Solution → **Methods**

2. Enable **Non-Iterative Time-Advancement**.

3. Under **Pressure-Velocity Coupling**, select a scheme from the drop-down list:

   - For single phase flows: You can select either the **Fractional Step** or **PISO** scheme. For the **PISO** scheme, you can set the value for the **Neighbor Correction**. Skewness correction is performed automatically.

   - For Eulerian multiphase flows: Select the **Phase Coupled SIMPLE**. This is the only pressure scheme available for NITA multiphase simulations.

Once you enable **Non-Iterative Time-Advancement**, you can set the parameters that control the sub-iterations for individual equations under **Non-Iterative Solver Controls** in the **Advanced Solution Controls** dialog box (Figure 28.6: The Advanced Solution Controls Dialog Box for the Pressure-Based Segregated Non-Iterative Solver (p. 1567)). See User Inputs (p. 1566) for details.

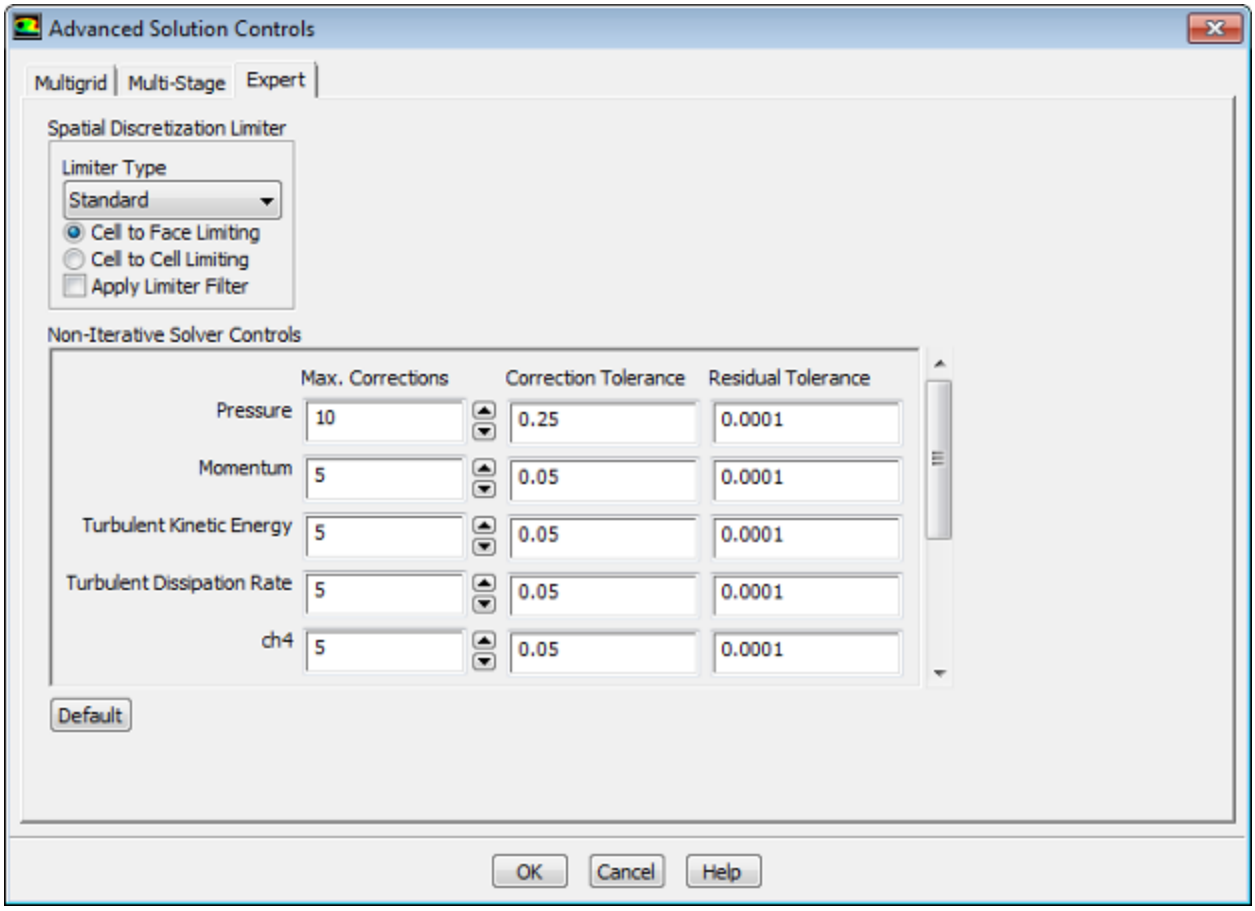
28.3.3.1. User Inputs

You can modify the non-iterative solution controls in the Advanced Solution Controls Dialog Box (p. 2513) (Figure 28.6: The Advanced Solution Controls Dialog Box for the Pressure-Based Segregated Non-Iterative Solver (p. 1567)).

Solution → **Controls** → **Advanced...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.6: The Advanced Solution Controls Dialog Box for the Pressure-Based Segregated Non-Iterative Solver



Under **Non-Iterative Solver Controls**, there are several parameters that control the sub-iterations for the individual equations.

The sub-iterations for an equation stop when the total number of sub-iterations exceeds the value specified for **Max. Corrections**, regardless of whether or not the convergence criteria (described below) are met.

The sub-iterations for an equation end when the ratio of the residuals at the current sub-iteration and the first sub-iteration is less than the value specified in the **Correction Tolerance** field. You can monitor the details of the sub-iteration convergence by looking at the AMG solver performance (that is, setting the **Verbosity** field in the **Multigrid** tab in the **Advanced Solution Controls** dialog box to **1**). Be sure to pay attention to the residuals for the current sub-iteration (that is, the residual for the 0-th AMG cycle at the current sub-iteration) and the initial residual of the time step (that is, the residual for the 0-th AMG cycle of the first sub-iteration). The ratio of these two residuals is what is controlled by the **Correction Tolerance** field. These two residuals are also the residuals plotted when using the **Residual Monitor** panel and reported in the ANSYS Fluent console at the end of a time step. Note that the residuals reported at the end of a time step can be scaled or unscaled, depending on the settings in the **Residual Monitor** dialog box. The residuals reported when monitoring the AMG solver performance are always unscaled.

For each interim sub-iteration, the AMG cycles continue until the usual AMG termination criteria (0.1 by default, and set in the **Multigrid** tab) are met. However, for the last sub-iteration (that is, either when the maximum number of sub-iterations are reached or when the correction tolerance is satisfied), the AMG cycles continue until the ratio of the residual at the current cycle to the initial residual (the residual

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

for the 0-th AMG cycle of the first sub-iteration of the time step) drops below the value specified for **Residual Tolerance**. You may want to adjust the **Residual Tolerance**, depending on the time step selected. The default **Residual Tolerance** should be well suited for moderate time steps (that is, for cell CFL numbers of 1 to 10). Note that you can display the cell CFL numbers for unsteady problems by selecting **Cell Courant Number** in the **Velocity...** category of all postprocessing dialog boxes. For very small time steps (cell CFL <<1), the diagonal dominance of the system is very high and the convergence should be driven further by reducing the **Residual Tolerance** value. For larger time steps (cell CFL >>1), it may be possible that the residual tolerance cannot be reached due to round-off errors, and unless the **Residual Tolerance** value is increased, AMG cycles can be wasted. Again, this can be monitored by monitoring the AMG solver performance.

The **Relaxation Factor** field defines the explicit relaxation (see Under-Relaxation of Variables in the Theory Guide) of variables between sub-iterations. The relaxation factors can be used to prevent the solution from diverging. They should be left at their default values of 1, unless divergence is detected. If the solution diverges:

- For single phase flows: You should first try to stabilize the solution by lowering the relaxation factors for pressure to 0.7–0.8, and by reducing the time step.
- For Eulerian multiphase flows: Consider lowering the explicit pressure relaxation factor to about 0.1. This may help when meshes are poor, because a poor mesh will significantly affect the pressure gradient, which in turn may generate large source terms in the governing equations resulting in poor convergence. You can set the under-relaxation factors for the other flow variables to higher values (0.8-1.0).

Expert Options

- To obtain diagnostics on convergence of sub-iterations for individual equations, use the following text command:

  ```
  solve/set/nita-expert-controls/set-verbosity

  verbosity [0] 1
  ```

  The output information may help you to judge whether the specified number of sub-iterations is sufficient to reach the correction tolerance and also to diagnose the source of the solution divergence.

- For NITA cases with **PISO** selected as a **Pressure-Velocity Coupling** scheme, you can enable coupling of the neighbor and skewness corrections with the following TUI command:

  ```
  solve/set/nita-expert-controls/skewness-neighbor-coupling

  enable skewness neighbor coupling for nita [no] yes
  ```

  For more information about this option, see Skewness - Neighbor Coupling in the *Fluent Theory Guide*.

Compatibility with Other ANSYS Fluent Models

The following is a list of models that are compatible with the non-iterative solver:

- Inviscid flow (excluding ideal gas)
- Laminar flow
- All models of turbulence (including LES and DES), except RSM

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- S2S radiation model
- Heat transfer
- Non-reacting species transport
- General compressible flows (most subsonic and some transonic applications)
- VOF multiphase model (most applications)
- Phase change (solidification and melting)
- Porous media model (isotropic resistance)
- Multiphase model with Multi-Fluid VOF
- Multiphase RSM turbulence model

The following is a list of models that are compatible with the non-iterative solver, but may result in some instabilities and inaccuracies for certain flow conditions:

- MDM
- Non-Newtonian fluids
- General compressible flows (aerospace supersonic applications)
- Reacting species and any type of combustion including PDF

The following is a list of models that are not compatible with the non-iterative solver:

- Mixture model
- Radiation models (except S2S)
- DPM, spark, and crevice models
- UDS transport
- Porous jump
- Porous media model (anisotropic resistance)
- RSM turbulence model

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Floating operating pressure

Important

The PRESTO! pressure interpolation scheme, when used with the non-iterative time-advancement solver, is less stable than in the case of the iterative time-advancement solver. As a consequence, smaller time steps may be required.

Important

As mentioned above, the default control settings are optimally designed to obtain a second-order solution. In order to save CPU time, in cases where transient accuracy is not a main concern (that is, first-order integration in time and space), or when NITA is used to converge toward a steady-state solution, you may want to set the **Max. Corrections** value to 1 in the **Advanced Solution Controls** dialog box (**Expert** tab) for all transport equations except pressure.

28.4. Density-Based Solver Settings

To use the density-based solver you must first select the solver type, and determine if the simulation is steady-state or transient from the **General** Task Page (see Choosing the Solver (p. 1553)). The density based solver settings are available mainly in two task pages: the Solution Methods Task Page (p. 2504) and the Solution Controls Task Page (p. 2508).

In the **Solution Methods** task page, you can select the following:

- Solution formulation type: Implicit or Explicit
- Flux Scheme type : Roe-FDS, AUSM or (Low Diffusion Roe-FDS)
- Flow equation and model equation spatial discretization accuracy
- For the steady-state solution method, you can select additional solution options to accelerate convergence
  - Pseudo transient solution method
  - Convergence acceleration for stretched meshes
- For the transient formulation you can select
  - 1st-Order implicit
  - 2nd-order implicit and for the explicit solver formulation, you can also select the explicit transient formulation

In the **Solution Controls** task page, you can select the following:

- For the density-based explicit solver, you will specify the Courant number, FAS multigrid level, and Residual smoothing
- For the density-based implicit solver, you will need to enter only the Courant number

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For both solver methods, you will enter the under-relaxation factors associated with other equations solved with the flow equations, such as equations of the turbulence model

The above options can be found in the following sections:
28.4.1. Changing the Courant Number
28.4.2. Convective Flux Types
28.4.3. Convergence Acceleration for Stretched Meshes (CASM)
28.4.4. Specifying the Explicit Relaxation
28.4.5. Turning On FAS Multigrid

28.4.1. Changing the Courant Number

For ANSYS Fluent's density-based solver, the main control over the time-stepping scheme is the Courant number (CFL). The time step is proportional to the CFL, as defined in Equation 21.81 in the Theory Guide.

Linear stability theory determines a range of permissible values for the CFL (that is, the range of values for which a given numerical scheme will remain stable). When you specify a permissible CFL value, ANSYS Fluent will compute an appropriate time step using Equation 21.81 in the Theory Guide. In general, taking larger time steps leads to faster convergence, so it is advantageous to set the CFL as large as possible (within the permissible range).

The stability limits of the density-based implicit and explicit formulations are significantly different. The explicit formulation has a more limited range and requires lower CFL settings than does the density-based implicit formulation. Appropriate choices of CFL for the two formulations are discussed below.

28.4.1.1. Courant Numbers for the Density-Based Explicit Formulation

Linear stability analysis shows that the maximum allowable CFL for the multi-stage scheme used in the density-based explicit formulation will depend on the number of stages used and how often the dissipation and viscous terms are updated (see Changing the Multi-Stage Scheme (p. 1588)). But in general, you can assume that the multi-stage scheme is stable for Courant numbers up to 2.5. This stability limit is often lower in practice because of nonlinearities in the governing equations.

The default CFL for the density-based explicit formulation is 1.0, but you may be able to increase it for some 2D problems. You should generally not use a value higher than 2.0.

If your solution is diverging, that is, if residuals are rising very rapidly, and your problem is properly set up and initialized, this is usually a good sign that the Courant number must be lowered. Depending on the severity of the startup conditions, you may need to decrease the CFL to a value as low as 0.1 to 0.5 to get started. Once the startup transients are reduced you can start increasing the Courant number again.

28.4.1.2. Courant Numbers for the Density-Based Implicit Formulation

Linear stability theory shows that the density-based implicit formulation is unconditionally stable. However, as with the explicit formulation, nonlinearities in the governing equations will often limit stability.

The default CFL for the density-based implicit formulation is 5.0. It is often possible to increase the CFL to 10, 20, 100, or even higher, depending on the complexity of your problem. You may find that a lower CFL is required during startup (when changes in the solution are highly nonlinear), but it can be increased as the solution progresses.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

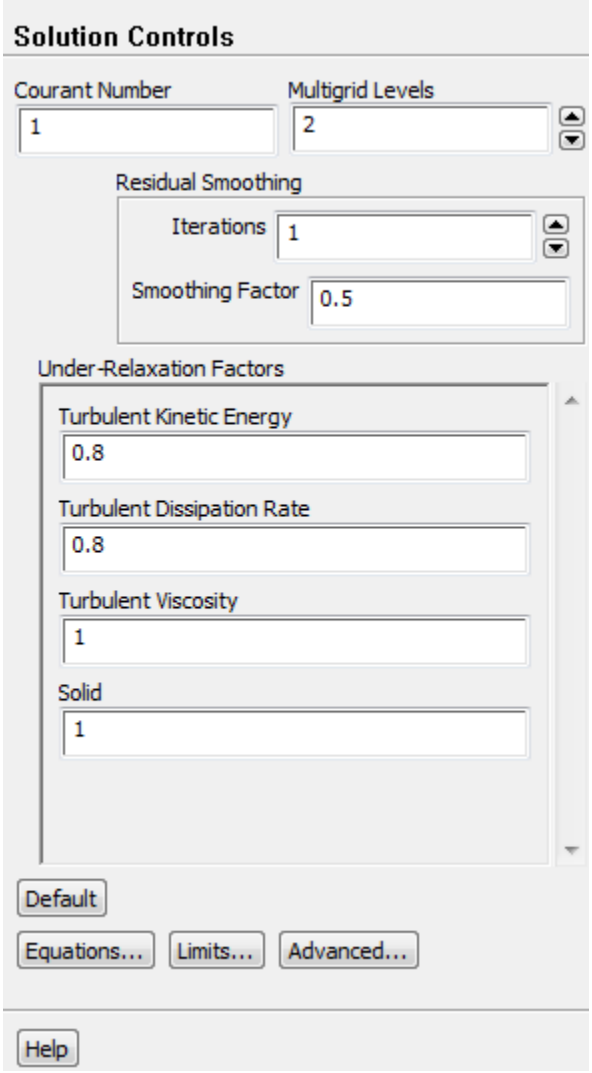
The coupled AMG solver has the capability to detect divergence of the multigrid cycles within a given iteration. If this happens, it will automatically reduce the CFL and perform the iteration again, and a message will be printed to the screen. Five attempts are made to complete the iteration successfully. Upon successful completion of the current iteration the CFL is returned to its original value and the iteration procedure proceeds as required.

28.4.1.3. User Inputs

The Courant number is set in the Solution Controls Task Page (p. 2508) (Figure 28.7: The Solution Controls Task Page for the Density-Based Explicit Formulation (p. 1572)).

Solution → Controls

Figure 28.7: The Solution Controls Task Page for the Density-Based Explicit Formulation



Enter the value for **Courant Number**.

When you select **Explicit** from the **Formulation** drop-down list in the Solution Methods Task Page (p. 2504), ANSYS Fluent will automatically set the **Courant Number** to 1; when you select **Implicit** from the **Formulation** drop-down list, the **Courant Number** will be changed to 5 automatically.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.4.2. Convective Flux Types

Three convective flux types exist when using the density-based solver:

- Roe flux-difference splitting (Roe-FDS)
- Advection Upstream Splitting Method (AUSM)
- Low diffusion Roe flux-difference splitting (Low Diffusion Roe-FDS)

Roe-FDS splits the fluxes in a manor that is consistent with their corresponding flux method eigenvalues. It is the default and is recommended for most cases.

AUSM provides exact resolution of contact and shock discontinuities and it is less susceptible to Carbuncle phenomena.

Low diffusion Roe-FDS is available in special circumstances when the LES viscous model is enabled and when time-implicit formulation is used in the density-based solvers. It reduces the dissipation in LES calculations. Low diffusion Roe-FDS should be used only for subsonic flows.

28.4.2.1. User Inputs

The convective fluxes are selected from the **Flux Type** drop-down list in the **Solution Methods** task page.

Solution → **Methods**

Select **Roe-FDS**, **AUSM**, or if the LES viscous model is enabled with the time-implicit formulation, **Low Diffusion Roe-FDS** will be available.

28.4.3. Convergence Acceleration for Stretched Meshes (CASM)

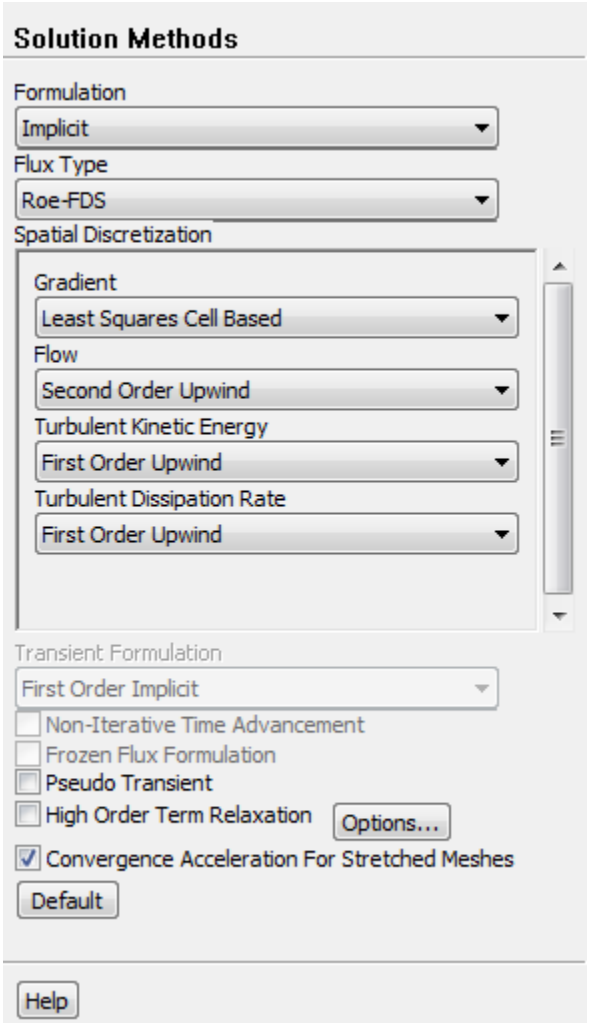
When using the density-based solver with the implicit solution formulation in steady-state you can accelerate the convergence of your solution on highly-stretched and anisotropic meshes (like the one used when modeling external aerodynamic problems) by selecting the **Convergence Acceleration For Stretched Meshes** in the **Solution Methods** task page. For further information and theoretical background on this solution acceleration option, see Convergence Acceleration for Stretched Meshes in the Theory Guide. The **Convergence Acceleration For Stretched Meshes** option provides an optimum solution convergence of the implicit solution method.

To apply convergence acceleration for stretched meshes, perform the following:

1. Specify the solver options by selecting **Density-Based** and **Steady** in the **General** task page.
2. Select **Implicit** from the **Formulation** drop-down list and enable **Convergence Acceleration For Stretched Meshes** in the **Solution Methods** task page (Figure 28.8: The Solution Methods Task Page for the Density-Based Implicit Formulation (p. 1574)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.8: The Solution Methods Task Page for the Density-Based Implicit Formulation



Extra settings for CASM can be set using the following text command:

`solve → set → convergence-acceleration-for-stretched-meshes/`

Enter `yes` in response to the `Use convergence acceleration for stretched meshes (CASM)?` question. You will also be asked for a cut-off on the CFL value multiplier. By default this value is set to 100. Typically, you do not need to adjust this value. But if convergence difficulties are encountered and reduction of the CFL value alone does not help improve convergence, then it is advisable to reduce this CFL multiplier cut-off to a lower value (for example from 100 to 50, 20 or 10).

The use of CASM can typically give a much faster convergence over the standard solution method. In general, when using CASM, you do not need to specify a very large CFL value as you do with the standard solution method. A CFL value between 5 and 10 is typically used for converging most flow problems. When the **Convergence Acceleration For Stretched Meshes** option is selected, the solver will run when appropriate with a variable local CFL value proportional to the cell aspect ratios. Therefore, when the cell aspect ratio nears unity (typically far from walls), the local cell CFL value will be the same as the value that you supplied. However, as the cell is stretched and the cell aspect ratio increases (near walls), the local cell CFL value will be multiplied by the cell aspect ratio value. This is true until the cell stretching is beyond the multiplier cut-off value specified using the text command. The proportional change in CFL value on highly stretched cells helps accelerate the solution especially on highly packed and stretched meshes like the one used in modeling external flow problems.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the cell aspect ratio is selected by default, the density based implicit solver will operate with an explicit relaxation of 0.5 which can be adjusted from the `Explicit underrelaxation value` entry in `solve → set → expert`

The convergence of the solution is mainly controlled by adjusting the CFL value. Therefore, if convergence problems are encountered, lowering the CFL value will help improve the convergence. Additional solution parameters to be adjusted for more conservative solution settings are:

1. lowering the density-based implicit solver explicit relaxation (`solve → set → expert`)
2. adjusting the CFL multiplier cut-off value to a lower value (`solve → set → convergence-acceleration-for-stretched-meshes/`)

This solution convergence method is very aggressive. Therefore it is of paramount importance to start with a good initial guess especially if you start with second order spatial discretization. To get a good starting solution with the guess you provide, you are advised to use the full multi-grid initialization method (see Full Multigrid (FMG) Initialization (p. 1595)).

Note

When selecting the **Convergence Acceleration For Stretched Meshes** option then the **Pseudo-Transient** solution method will not be available. You can either use **Convergence Acceleration For Stretched Meshes** or the **Pseudo-Transient** solution method. These two options cannot be used at the same time. Both methods help in obtaining faster convergence on anisotropic meshes. But one method requires that you enter a CFL value, while the other requires that you enter a pseudo-time step value to march the solution to convergence. It is up to you to select the method with which you are most comfortable.

Convergence Acceleration For Stretched Meshes shows an advantage over the standard solution method, particularly with stretched meshes with low Y+ values (near unity). The use of **Convergence Acceleration For Stretched Meshes** results in an alteration of the numerical dissipation of the selected flux scheme. This change may slightly impact the monitored loading level if compared with the solution obtained without the use of this option.

28.4.4. Specifying the Explicit Relaxation

To improve the convergence to steady-state for some flow cases when using the density-based implicit solver you can specify the explicit relaxation using the following text command:

`solve → set → expert`

Enter a value between `0` and `1` in response to the `Explicit relaxation value` prompt.

For more information about explicit relaxation, see Under-Relaxation of Variables in the Theory Guide.

28.4.5. Turning On FAS Multigrid

As discussed in Multigrid Method in the Theory Guide, FAS multigrid is an optional component of the density-based explicit formulation, while AMG multigrid is always on, by default for the density-based implicit formulation. Since nearly all density-based explicit calculations will benefit from the use of the FAS multigrid convergence accelerator, you should generally set a nonzero number of coarse grid levels before beginning the calculation. For most problems, this will be the only FAS multigrid parameter you

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

will need to set. Should you encounter convergence difficulties, consider applying one of the methods discussed in Setting FAS Multigrid Parameters (p. 1583).

Important

Note that you cannot use FAS multigrid with explicit time stepping (described in Temporal Discretization in the Theory Guide) because the coarse grid corrections will destroy the time accuracy of the fine grid solution.

28.4.5.1. Setting Coarse Grid Levels

As discussed in Full-Approximation Storage (FAS) Multigrid in the Theory Guide, FAS multigrid solves on successively coarser grids and then transfers corrections to the solution back up to the original fine grid, thereby increasing the propagation speed of the solution and speeding convergence. The most basic way you can control the multigrid solver is by specifying the number of coarse grid levels to be used.

As explained in Full-Approximation Storage (FAS) Multigrid in the Theory Guide, the coarse grid levels are formed by agglomerating a group of adjacent "fine" cells into a single "coarse" cell. The optimal number of grid levels is therefore problem-dependent. For most problems, you can start out with 4 or 5 levels. For large 3D problems, you may want to add more levels (although memory restrictions may prevent you from using more levels, since each coarse grid level requires additional memory). If you believe that multigrid is causing convergence trouble, you can decrease the number of levels.

If ANSYS Fluent reaches a coarse grid with one cell before creating as many levels as you requested, it will simply stop there. That is, if you request 5 levels, and level 4 has only 1 cell, ANSYS Fluent will create only 4 levels, since levels 4 and 5 would be the same.

To specify the number of grid levels you want, set the number of **Multigrid Levels** in the Solution Controls Task Page (p. 2508) (Figure 28.7: The Solution Controls Task Page for the Density-Based Explicit Formulation (p. 1572)).

Solution → **Controls**

You can also set the **Max Coarse Levels** under **FAS Multigrid Controls** in the **Multigrid** tab in the Advanced Solution Controls Dialog Box (p. 2513).

Changing the number of coarse grid levels in the **Solution Controls** task page will automatically update the number shown in the **Multigrid** tab in the Advanced Solution Controls Dialog Box (p. 2513).

Coarse grid levels are created when you first begin iterating. If you want to check how many cells are in each level, request one iteration and then click **Info** and select **Size** in the **Setting Up Domain** ribbon tab (**Mesh** group box) (as described in Mesh Size (p. 197)) to list the size of each grid level. If you are satisfied, you can continue the calculation; if not, you can change the number of coarse grid levels and check again.

For most problems, you will not need to modify any additional multigrid parameters once you have settled on an appropriate number of coarse grid levels. You can simply continue your calculation until convergence.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.4.5.2. Using Residual Smoothing to Increase the Courant Number

In the density-based explicit formulation, implicit residual smoothing (or averaging) is a technique that can be used to reduce the time step restriction of the solver, thereby allowing the Courant number to be increased. The implicit smoothing is implemented with an iterative Jacobi method, as described in Implicit Residual Smoothing in the Theory Guide. Solution Controls Task Page (p. 2508).

Solution → Controls

By default, the number of **Iterations** for **Residual Smoothing** is set to zero, indicating that residual smoothing is disabled. If you increase the **Iterations** counter to 1 or more, you can enter the **Smoothing Factor**. A smoothing factor of 0.5 with 2 passes of the Jacobi smoother is usually adequate to allow the Courant number to be doubled.

28.5. Setting Algebraic Multigrid Parameters

As mentioned earlier, in most cases the multigrid solver will not require any special attention from you. If, however, you have convergence difficulties or you want to minimize the overall solution time by using more aggressive settings, you can monitor the multigrid solver and modify the parameters to improve its performance. (The instructions below assume that you have already begun calculations, since there is no need to monitor the solver if you do not fit into one of the two categories above.)

To determine whether your convergence difficulties can be alleviated by modifying the multigrid settings, you will check if the requested residual reduction is obtained on each grid level. To minimize solution time, you will check to see if switching to a more powerful cycle will result in overall reduction of work by the solver.

By default, the flexible cycle is used for all equations except pressure correction, which uses a V cycle. Typically, for a flexible cycle only a few (5–10) relaxations will be performed at the finest level and no coarse levels will be visited. In some cases one or two coarse levels may be visited. If the maximum number of fine level relaxations is not sufficient, you may want to increase the maximum number (as described in Flexible Cycle Parameters (p. 1581)) or switch to a V cycle (as described in Specifying the Multigrid Cycle Type (p. 1578)).

In the pressure-based segregated algorithm, the pressure correction uses a V cycle by default. If the maximum number of cycles (30 by default) is not sufficient, you can switch to a W cycle (using the **Multigrid** tab in the Advanced Solution Controls Dialog Box (p. 2513), as described in Specifying the Multigrid Cycle Type (p. 1578)). Note that for the parallel solver, efficiency may deteriorate with a W cycle.

If you are using the parallel solver, you can try increasing the maximum number of cycles by increasing the value of **Max Cycles** in the **Multigrid** tab, under **Fixed Cycle Parameters**.

In the pressure-based coupled algorithm and the density-based implicit formulation, there is no pressure correction. Instead, there is a flow correction, which by default uses the F cycle. The density-based explicit formulation uses the V cycle as the default flow correction.

Solution → Controls → Advanced...

For additional information, see the following sections:

28.5.1. Specifying the Multigrid Cycle Type
28.5.2. Setting the Termination and Residual Reduction Parameters
28.5.3. Setting the AMG Method and the Stabilization Method
28.5.4. Additional Algebraic Multigrid Parameters

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.5.5. Setting FAS Multigrid Parameters

28.5.1. Specifying the Multigrid Cycle Type

By default, the V cycle is used for the pressure equation in the pressure-based segregated algorithm and the flexible cycle is used for all other equations with the exception that the F cycle is used for energy. In the pressure-based coupled algorithm, the F cycle is the default for the coupled flow equations and for the energy equation. All other scalar equations use the flexible cycle by default. In the density-based implicit formulation, the F cycle is default for the flow correction. The V cycle is default for the flow correction in the density-based explicit formulation. For both density-based formulations the flexible cycle is used for the scalar equations. (See Multigrid Cycles in the Theory Guide for a description of these cycles.) To change the cycle type for an equation, you will use the top portion of the **Multigrid** tab in the Advanced Solution Controls Dialog Box (p. 2513) (Figure 28.9: The Multigrid Tab for the Flexible Cycle (p. 1582)).

For each equation, you can choose **Flexible**, **V-Cycle**, **W-Cycle**, or **F-Cycle** in the adjacent drop-down list.

28.5.2. Setting the Termination and Residual Reduction Parameters

When you use the flexible cycle for an equation, you can control the multigrid performance by modifying the **Termination** and/or **Restriction** criteria for that equation at the top of the **Multigrid** tab in the Advanced Solution Controls Dialog Box (p. 2513) (Figure 28.9: The Multigrid Tab for the Flexible Cycle (p. 1582)).

Solution → **Controls** → **Advanced...**

The **Restriction** criterion is the residual reduction tolerance, β in Equation 21.122 in the Theory Guide. This parameter dictates when a coarser grid level must be visited (due to insufficient improvement in the solution on the current level). With a larger value of β, coarse levels will be visited less often (and vice versa). The **Termination** criterion, α in Equation 21.123 in the Theory Guide, governs when the solver should return to a finer grid level (that is, when the residuals have improved sufficiently on the current level).

For the V, W, or F cycle, the **Termination** criterion determines whether or not another cycle should be performed on the finest (original) level. If the current residual on the finest level does not satisfy Equation 21.123 in the Theory Guide, and the maximum number of cycles has not been performed, ANSYS Fluent will perform another multigrid cycle. (The **Restriction** parameter is not used by the V, W, and F cycles.)

28.5.3. Setting the AMG Method and the Stabilization Method

You can use the **Multigrid** tab in the Advanced Solution Controls Dialog Box (p. 2513) (Figure 28.9: The Multigrid Tab for the Flexible Cycle (p. 1582)) to choose between two AMG solvers: aggregative or selective. The aggregative AMG (AAMG) is the default solver that was used in previous versions of ANSYS Fluent. The selective AMG (SAMG) solver is available only for scalar equations, and is not available in parallel ANSYS Fluent. These two solvers differ in the way the grids are coarsened and in their interpolation method.

The AAMG solver [128] (p. 2894) builds coarse levels by grouping fine level cells to make coarse level cells, and uses piecewise constant interpolation. The SAMG solver [114] (p. 2893) builds coarse levels by

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

selecting some of the fine level cells for solution on the coarse level, and tries to approximate the use of linear interpolation.

Due to its use of more accurate interpolation, SAMG has a better convergence rate than AAMG but has a more expensive setup phase. For this reason, AAMG is usually faster if you are only converging one order of magnitude, while SAMG is faster if using a tight multigrid convergence tolerance. SAMG is a good choice for multiphase granular flow problems where a tight convergence tolerance on the pressure equation can be used to avoid volume imbalance errors in the volume fraction equations.

SAMG has advantages in solving problems with strongly varying (anisotropic) diffusive coefficients, which occurs in problems with porous media, conduction with anisotropic thermal conductivities, and multiphase problems. In some cases, using SAMG allows up to a 20% reduction in the number of external iterations for unsteady water-air turbulent flow in bubble columns, and allows increasing the VOF under-relaxation factor in phase separators from 0.2 (when used with AAMG) to 1.

In the **Multigrid** tab (Figure 28.9: The Multigrid Tab for the Flexible Cycle (p. 1582)), you can also choose a stabilization method. If desired, you can choose the bi-conjugate gradient stabilized method [9] (p. 2887) (**BCGSTAB**) option or recursive projection method [104] (p. 2892) (**RPM**) in order to improve the convergence of the linear solver. BCGSTAB can be preconditioned by any of the AMG solvers and provides stabilization for them whereas RPM stabilizes the AAMG solver.

ANSYS Fluent usually builds diagonally dominant matrices for the linear solver. However, this is not always possible. A linear system with highly dominant off-diagonal coefficients may occur during discretization of complex physical models such as multiphase cavitation. Using the **BCGSTAB** or **RPM** option in such cases can be helpful. In addition, the AMG convergence in parallel can be improved using the **BCGSTAB** option with AMG.

If you are using the pressure-based segregated solver and the flow is incompressible, an additional stabilization method for the algebraic multigrid solver will appear in the **Stabilization Method** drop-down list. This is the conjugate gradient method, or **CG** [9] (p. 2887) and will be available only for the pressure equation. This method is typically used in conjunction with an AMG preconditioner (any available AMG method). The CG formulation requires a symmetric system matrix. Such matrices result from the finite volume discretization of steady or transient elliptic operators, such as the pressure correction equation in the incompressible case. The CG method provides a useful extension to BCGSTAB and RPM, since it reduces the memory requirements and the number of floating point operations, especially when compared to BCGSTAB. In addition, in transient pressure-based segregated solvers, as typically used in LES and/or NITA simulations, the solution of the pressure correction equation constitutes a significant share of the overall computational effort.

28.5.4. Additional Algebraic Multigrid Parameters

There are several additional parameters that control the algebraic multigrid solver, but there will usually be no need to modify them. These additional scalar and coupled parameters are all contained in the **Multigrid** tab in the Advanced Solution Controls Dialog Box (p. 2513) (Figure 28.9: The Multigrid Tab for the Flexible Cycle (p. 1582)).

Solution → **Controls** → **Advanced...**

Important

When using the density-based explicit formulation or the pressure-based solver with any of the segregated algorithms, described in Pressure-Velocity Coupling in the Theory Guide and

Choosing the Pressure-Velocity Coupling Method (p. 1561), only **Scalar Parameters** are set in the **Multigrid** tab. If you use the density-based implicit or the pressure-based coupled algorithm, described in Coupled Algorithm in the Theory Guide, then you can set the **Coupled Parameters**.

28.5.4.1. Fixed Cycle Parameters

For the fixed (V, W, and F) multigrid cycles, you can control the number of pre- and post-relaxations (β_1 and β_3 in Multigrid Cycles in the Theory Guide). **Pre-Sweeps** sets the number of relaxations to perform before moving to a coarser level. **Post-Sweeps** sets the number to be performed after coarser level corrections have been applied. Normally, under **Scalar Parameters**, one post-relaxation is performed and no pre-relaxations are done (that is, $\beta_3=1$ and $\beta_1=0$), but in rare cases, you may need to increase the value of β_1 to 1 or 2. Under **Coupled Parameters**, three post-relaxations are performed by default with no pre-relaxations. If you are using the pressure-based coupled solver for a steady simulation with pseudo-transient enabled, three post-relaxations are performed under **Scalar Parameters** also.

Important

- If you are using AMG with V-cycle to solve an energy equation with a solid conduction model presented with anisotropic or very high conductivity coefficient, there is a possibility of divergence with a default post-relaxation sweep of 1. In such cases you should increase the post-relaxation sweep (to say 2) in the AMG section for better convergence when using the pressure-based segregated algorithms.
- It is recommended that you use the Fixed F-cycle for the energy equation when running parallel ANSYS Fluent.

28.5.4.2. Coarsening Parameters

For all multigrid cycle types, you can control the maximum number of coarse levels (**Max Coarse Levels** under **Scalar** or **Coupled Parameters**) that will be built by the multigrid solver.

Sets of coarser simultaneous equations are built until the maximum number of levels has been created, or the coarsest level has only 3 equations. Each level has about half as many unknowns as the previous level, so coarsening until there are only a few cells left will require about as much total coarse-level coefficient storage as was required on the fine mesh. Reducing the maximum coarse levels will reduce the memory requirements, but may require more iterations to achieve a converged solution. Setting **Max Coarse Levels** to 0 turns off the algebraic multigrid solver.

Another coarsening parameter you can control is the increase in coarseness on successive levels. The **Coarsen by** parameter specifies the number of fine grid cells that will be grouped together to create a coarse grid cell. The algorithm groups each cell with its strongest neighbor, then groups the cell and its strongest neighbor with the neighbor's strongest neighbor, continuing until the desired coarsening is achieved. Typical values for the scalar parameters are in the range from 2 to 10, with the default value of 2 for the Gauss-Seidel smoother giving the best performance, but also the greatest memory use. For coupled parameters and scalar parameters when using the pressure-based coupled solver with pseudo-transient enabled, the default values of 4 (for 2D) and 8 (for 3D) for the ILU smoother give the best

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

performance. You should not adjust this parameter unless you need to reduce the memory required to run a problem.

Important

Depending on the smoother type, Gauss-Seidel or ILU, the **Coarsen by** and **Post-Sweeps** settings should be changed as follows when selecting the non-default smoother type:

ILU
: **Post-Sweeps**=3 and **Coarsen by** = 8

Gauss-Seidel
: **Post-Sweeps**=1 and **Coarsen by** = 2

By default, a cell's strongest neighbor is identified based on the magnitude of its interaction with the cell in question. Alternatively, you can enable **Laplace Coarsening** under **Scalar Parameters** or **Coupled Parameters** in the **Multigrid** tab of the **Advanced Solution Controls** dialog box. The Laplace coarsening option will use Laplace coefficients to evaluate neighbor strength when grouping cells for coarsening. This may improve stability in some cases because the coarser levels will not change as the solution evolves. It may also reduce computation time, particularly at high core counts, because the coarse levels don't need to be recreated every iteration.

For coupled equations, a **Conservative Coarsening** AMG option is enabled by default. This option improves convergence for difficult problems by tuning multigrid coarsening based on coefficient strengths and, in parallel computations, the partitioning.

28.5.4.3. Smoother Types

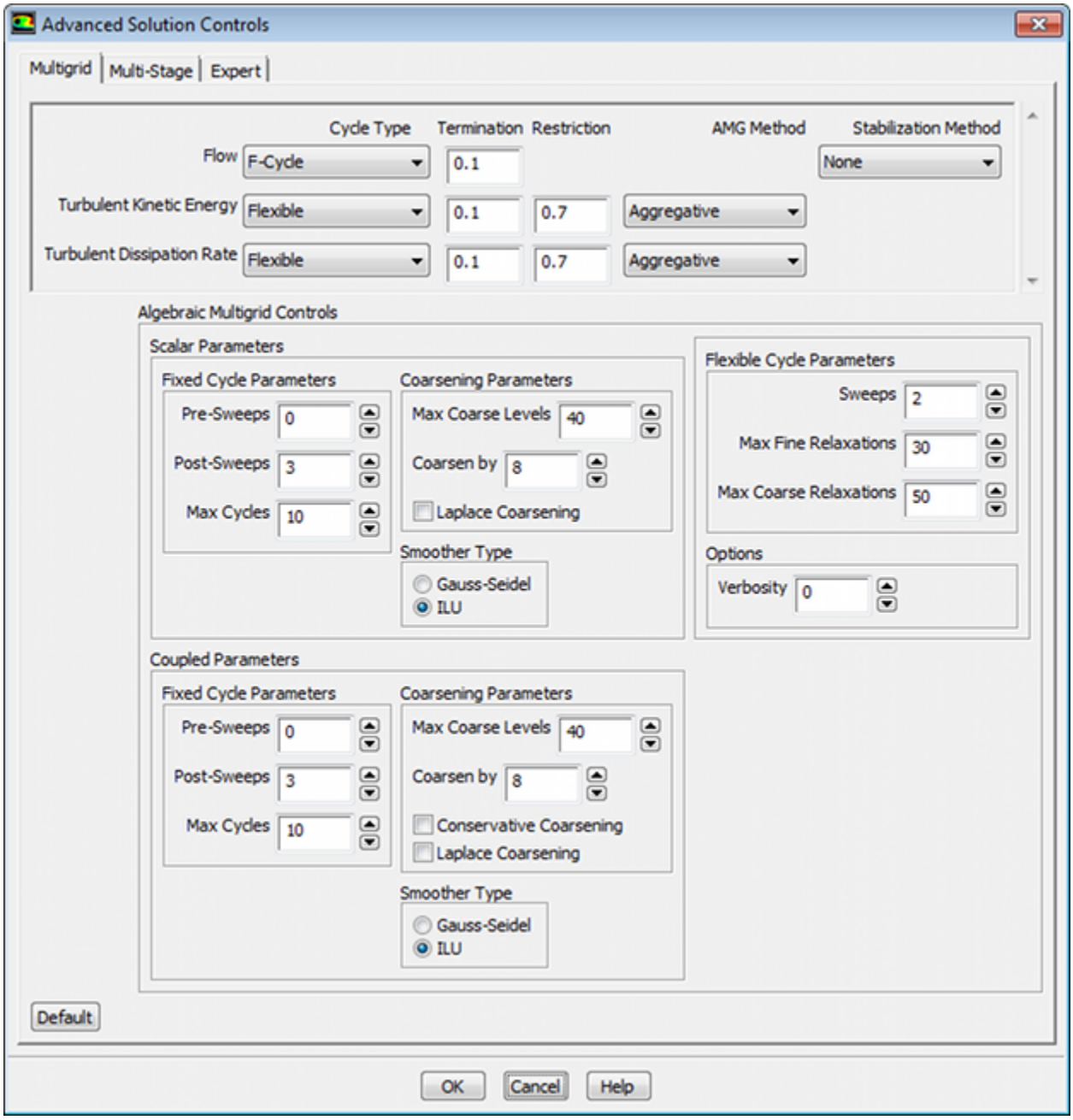
Two smoother types are available for scalar and coupled parameters. **Gauss-Seidel** is the simplest smoother type and is recommended when using the pressure-based segregated algorithm. **ILU** is more CPU intensive, but has better smoothing properties for block-coupled systems such as the pressure-based coupled solver and the density-based implicit formulation. The default scalar **Smoother Type** is **Gauss-Seidel**, while the coupled **Smoother Type** is **ILU**. The **ILU** smoother is also used for scalar equations when using the coupled solver with pseudo-transient enabled. For more information about the two smoother types, see The Coupled and Scalar AMG Solvers in the Theory Guide.

28.5.4.4. Flexible Cycle Parameters

To change the maximum number of relaxations, increase or decrease the value of **Max Fine Relaxations** or **Max Coarse Relaxations** in the **Multigrid** tab in the Advanced Solution Controls Dialog Box (p. 2513) (Figure 28.9: The Multigrid Tab for the Flexible Cycle (p. 1582)) under **Flexible Cycle Parameters**.

Solution → **Controls** → **Advanced...**

Figure 28.9: The Multigrid Tab for the Flexible Cycle



28.5.4.5. Setting the Verbosity

The steps for monitoring the solver are as follows:

1. Set multigrid **Verbosity** to 1 or 2 in the **Multigrid** tab in the Advanced Solution Controls Dialog Box (p. 2513).

   Solution → **Controls** → **Advanced...**

2. Request a single iteration using the Run Calculation Task Page (p. 2537).

   Solution → **Run Calculation**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you set the verbosity to 2, the information printed in the ANSYS Fluent console for each equation will include the following:

- equation name
- equation tolerance (computed by the solver using a normalization of the source vector)
- residual value after each fixed multigrid cycle or fine relaxation for the flexible cycle
- number of equations in each multigrid level, with the zeroth level being the original (finest-level) system of equations

Note that the residual printed at cycle or relaxation 0 is the initial residual before any multigrid cycles are performed.

When verbosity is set to 1, only the equation name, tolerance, and residuals are printed.

A portion of a sample printout is shown below:

```
pressure correction equation:
tol. 1.2668e-05
 0 2.5336e+00
 1 4.9778e-01
 2 2.5863e-01
 3 1.9387e-01

 multigrid levels:
 0 918
 1 426
 2 205
 3  97
 4  45
 5  21
 6  10
 7  4
```

28.5.4.6. Returning to the Default Multigrid Parameters

If you change the multigrid parameters, but you then want to return to ANSYS Fluent's default settings, you can click the **Default** button in the **Multigrid** tab. ANSYS Fluent will change all settings to the defaults, and the **Default** button will become the **Reset** button. To get your settings back again, you can click the **Reset** button.

28.5.5. Setting FAS Multigrid Parameters

For most calculations, you will not need to modify any FAS multigrid parameters once you have set the number of coarse grid levels. If, however, you encounter convergence difficulties, you may consider the following suggested procedures.

Important

Recall that FAS multigrid is used only by the density-based explicit formulation.

28.5.5.1. Combating Convergence Trouble

Some problems may approach convergence steadily at first, but then the residuals will level off and the solution will "get stuck." In some cases (for example, long thin ducts), this convergence trouble may

be due to multigrid's slow propagation of pressure information through the domain. In such cases, you should turn off multigrid by setting **Multigrid Levels** to 0 in the Solution Controls Task Page (p. 2508).

28.5.5.2. "Industrial-Strength" FAS Multigrid

In some cases, you may find that your problem is converging, but at an extremely slow rate. Such problems can often benefit from a more aggressive form of multigrid, which will speed up the propagation of the solution corrections. For such problems, you can try the "industrial-strength" multigrid settings.

Important

These settings are very aggressive and assume that the solution information passed through the multigrid levels is somewhat accurate. For this reason, you should only attempt the procedure described here after you have performed enough iterations that the solution is off to a good start. Using "industrial-strength" multigrid too early in the calculation process—when the solution is far from correct—will not help convergence and may cause the calculation to become unstable, as very incorrect values are propagated quickly to the original grid. Note also that while these multigrid settings will usually reduce the total number of iterations required to reach convergence, they will greatly increase the computation time for each multigrid cycle. Thus the solver will be performing fewer but longer iterations.

The strategy employed is as follows:

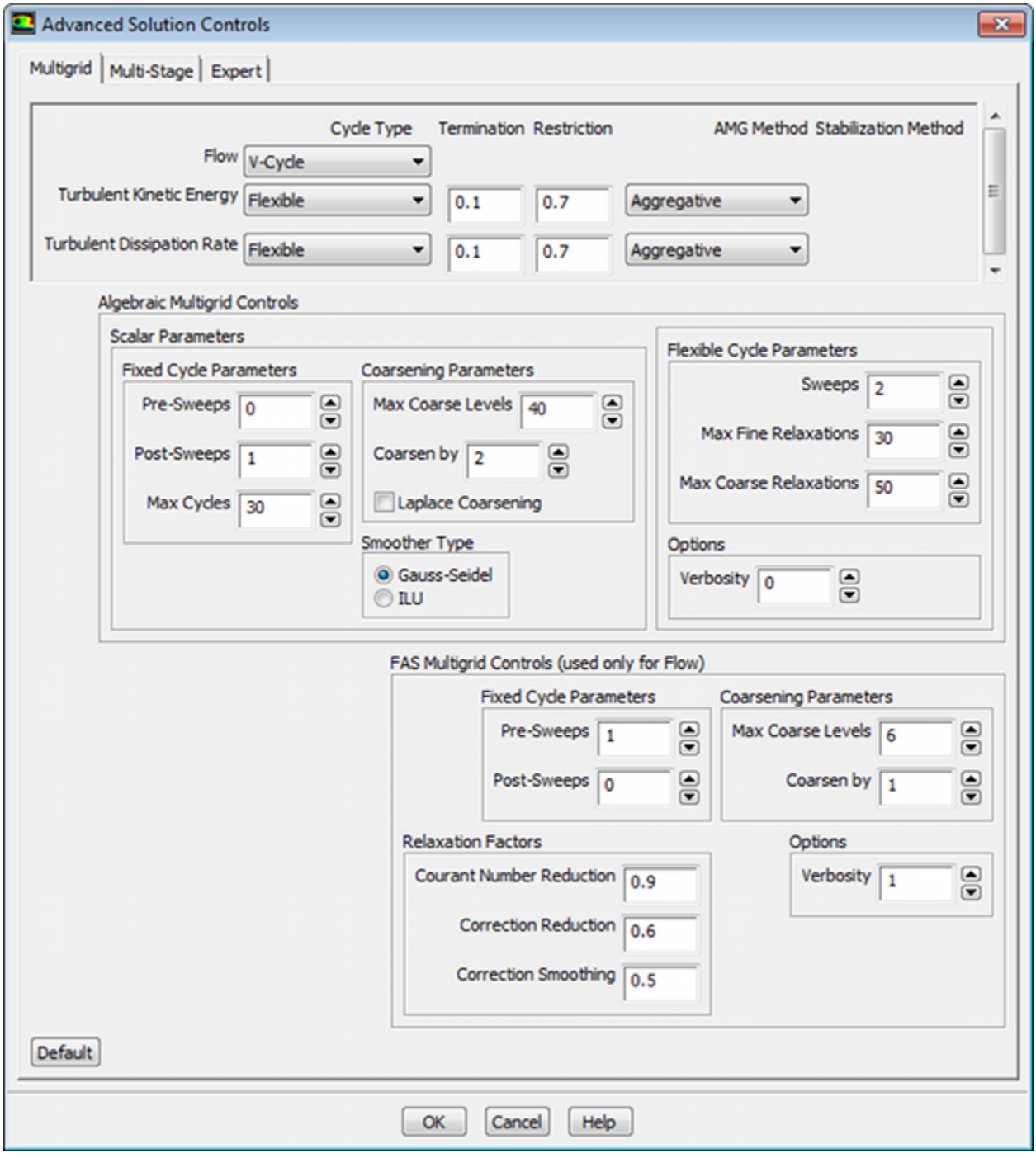
- Increase the number of iterations performed on each grid level before proceeding to a coarser level
- Increase the number of iterations performed on each grid level after returning from a coarser level
- Allow full correction transfer from one level to the next finer level, instead of transferring reduced values of the corrections
- Do not smooth the interpolated corrections when they are transferred from a coarser grid to a finer grid

You can set all of the parameters for this strategy under **FAS Multigrid Controls** in the **Multigrid** tab in the Advanced Solution Controls Dialog Box (p. 2513) (Figure 28.10: The Advanced Solution Controls Dialog Box (p. 1585)) and then continue the calculation.

Solution → **Controls** → **Advanced...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.10: The Advanced Solution Controls Dialog Box



Increasing the number of iterations performed on each grid level before proceeding to a coarser level (the value of β_1 described in Multigrid Cycles in the Theory Guide) will improve the solution passed from each finer grid level to the next coarser grid level. Try increasing the value of **Pre-Sweeps** (under **FAS Multigrid Controls**, *not* under **Algebraic Multigrid Controls**) to 10.

Increasing the number of iterations performed on each level after returning from a coarser level will improve the corrections passed from each coarser grid level to the next finer grid level. Errors introduced on the coarser grid levels can therefore be reduced before they are passed further up the grid hierarchy to the original grid. Try increasing the value of **Post-Sweeps** (under **FAS Multigrid Controls**, *not* under **Algebraic Multigrid Controls**) to 10.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

By default, the full values of the multigrid corrections are not transferred from a coarser grid to a finer grid; only 60% of the value is transferred. This prevents large errors from transferring quickly up to the original grid and causing the calculation to become unstable. It also prevents a "good" solution from propagating quickly to the original grid. However, by increasing the **Correction Reduction** to 1, you can transfer the full values from coarser to finer grid levels, speeding the propagation of the solution and, usually, the convergence as well. The **Species Correction Reduction** sets the factor by which to reduce the magnitude of the species corrections to stabilize the multigrid calculation. This item appears only when species transport is being modeled.

When the corrections on a coarse grid are passed back to the next finer grid level, the values are, by default, interpolated and then smoothed. Disabling the smoothing so that the actual value in a coarse grid cell is assigned to the fine grid cells that comprise it can also aid convergence. To disable smoothing, set the **Correction Smoothing** to 0. Large discontinuities between cells will be smoothed out implicitly as a result of the additional **Post-Sweeps** performed.

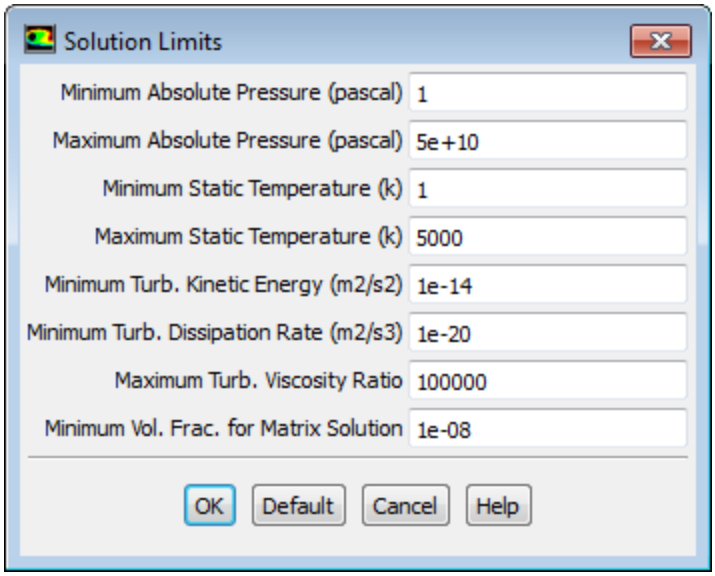
The **Courant Number Reduction** sets the factor by which to reduce the Courant number for coarse grid levels (that is, every level except the finest). Some reduction of time step (such as the default 0.9) is typically required because the stability limit cannot be determined as precisely on the irregularly shaped coarser grid cells.

28.6. Setting Solution Limits

In order to keep the solution stable under extreme conditions, ANSYS Fluent provides limits that keep the solution within an acceptable range. You can control these limits with the Solution Limits Dialog Box (p. 2511) (Figure 28.11: The Solution Limits Dialog Box (p. 1586)).

Solution → Controls → Limits...

Figure 28.11: The Solution Limits Dialog Box



Fluent applies limiting values for pressure, static temperature, and turbulence quantities. Fluent also applies a minimum volume fraction for the matrix solution in Eulerian multiphase simulations run on a double-precision solver. The purpose of these limits is to keep the absolute pressure or static temperature from becoming zero, negative, or excessively large during the calculation, and to keep the turbu-

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

lence quantities from becoming excessive. The purpose of the minimum volume fraction is to avoid singularity of the solution matrix when the volume fraction tends to zero.

Typically, you will not need to change the default solution limits. If pressure, temperature, or turbulence quantities are being reset to the limiting value repeatedly (as indicated by the appropriate warning messages in the console), you should check the dimensions, boundary conditions, and properties to be sure that the problem is set up correctly and try to determine why the variable in question is getting so close to zero or so large. You can use the "marking" feature (used to mark cells for adaption) to identify which cells have a value equal to the limit. (Use the Iso-Value Adaption Dialog Box (p. 2722), as described in Isovalue Adaption (p. 1691).) In very rare cases, you may need to change the solution limits, but only do so if you are sure that you understand the reason for the solver's unusual behavior. For example, you may know that the temperature in your domain will exceed 5000 K. Be sure that any temperature-dependent properties are appropriately defined for high temperatures if you increase the maximum temperature limit. Another rare example is when the actual volume fraction in the domain needs to go below a safe limit of 1e-08, such as in nucleation and dispersion of very low amounts of contaminants in a system. By reducing this limit you may encounter instability problems.

Important

The absolute pressure limit is enforced only for fluid materials that are modeled using the ideal-gas or one of the real-gas-* density methods.

For materials that use the compressible-liquid or a user-defined compressible fluid (liquid or gas) density method, a pressure limit is used to ensure that the density remains positive, but the actual pressure value itself is not limited.

For additional information, see the following sections:

28.6.1. Limiting the Values of Solution Variables
28.6.2. Adjusting the Positivity Rate Limit
28.6.3. Resetting Solution Limits

28.6.1. Limiting the Values of Solution Variables

The limiting minimum and maximum values for absolute pressure are shown in the **Minimum** and **Maximum Absolute Pressure** fields. If the ANSYS Fluent calculation predicts a value less than the **Minimum Absolute Pressure** or greater than the **Maximum Absolute Pressure**, the corresponding limiting value will be used instead. Similarly, the **Minimum** and **Maximum Temperature** are limiting values for energy calculations.

The **Minimum Turb. Kinetic Energy**, **Minimum Turb. Dissipation Rate**, and the **Maximum Turb. Viscosity Ratio** are limiting values for turbulent calculations. If the calculation predicts a value for k or ε that is less than the appropriate limiting value (that is, **Minimum Turb. Kinetic Energy** or **Minimum Turb. Dissipation Rate**, respectively), then the limiting value will be used instead. For the viscosity ratio limit, ANSYS Fluent uses the limiting maximum value of turbulent viscosity ($C_\mu k^2 / \varepsilon$) in the flow field relative to the laminar viscosity. If the ratio calculated by ANSYS Fluent exceeds the limiting value, the ratio is set to the limiting value by limiting ε to the necessary value.

For Eulerian multiphase simulations run on a double-precision solver, if the ANSYS Fluent calculation predicts a volume fraction value less than the **Minimum Vol. Frac. for Matrix Solution**, the limiting value will be used instead in the matrix solution.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.6.2. Adjusting the Positivity Rate Limit

In ANSYS Fluent's density-based solver, the rate of reduction of temperature is controlled by the **Positivity Rate Limit**. The default value of 0.2, for example, means that temperature is not allowed to decrease by more than 20% of its previous value from one iteration to the next. If the temperature change exceeds this limit, the time step in that cell is reduced to bring the change back into range and a "time step reduced" warning is printed. (This reduced time step will be used for the solution of all variables in the cell, not just for temperature.) Rapid reduction of temperature is an indication that the temperature may become negative. Repeated "time step reduced" warnings should alert you that something is wrong in your problem setup. (If the warning messages stop appearing, the calculation may have "recovered" from the time-step reduction.)

Important

For high-speed flow, if your solution is diverging particularly for the energy equation, then lowering this limit to 0.05 or 0.02 might help in overcoming divergence.

28.6.3. Resetting Solution Limits

If you change and save the value of one of the solution limits, but you then want to return to the default limits set by ANSYS Fluent, you can reopen the Solution Limits Dialog Box (p. 2511) and click the **Default** button. ANSYS Fluent will change the values to the defaults and the **Default** button will become the **Reset** button. To get your values back again, you can click the **Reset** button.

28.7. Setting Multi-Stage Time-Stepping Parameters

The most common parameter you will change to control the multi-stage time-stepping scheme is the Courant number. Instructions for modifying the Courant number are presented in Changing the Courant Number (p. 1571). The **Multi-Stage** tab is accessible from the **Advanced Solution Controls** dialog box when using the density-based explicit formulation.

For additional information, see the following section:
28.7.1. Changing the Multi-Stage Scheme

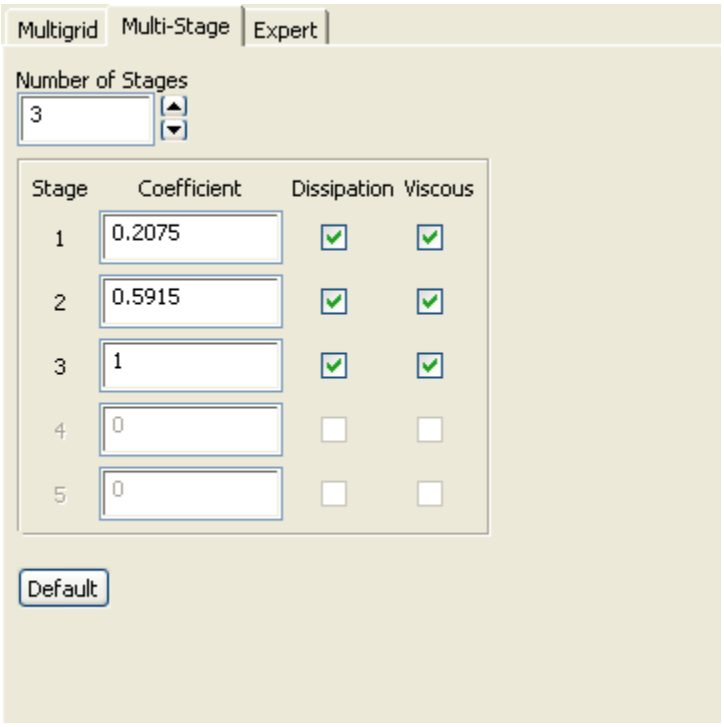
28.7.1. Changing the Multi-Stage Scheme

It is possible to make several changes to the multi-stage time-stepping scheme itself. You can change the number of stages and set a new multi-stage coefficient for each stage. You can also control whether or not dissipation and viscous stresses are updated at each stage. These changes are made in the **Multi-Stage** tab in the Advanced Solution Controls Dialog Box (p. 2513) (Figure 28.12: The Multi-Stage Tab (p. 1589)).

Solution → **Controls** **Advanced...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.12: The Multi-Stage Tab



Important

You should not attempt to make changes to ANSYS Fluent's multi-stage scheme unless you are very familiar with multi-stage schemes and are interested in trying a different scheme found in the literature.

28.7.1.1. Changing the Coefficients and Number of Stages

By default, the ANSYS Fluent multi-stage scheme uses 3 stages for steady-state solutions with coefficients of 0.2075, 0.5915, and 1.0, and 4 stages for unsteady solutions with coefficients of 0.25, 0.3333, 0.5, and 1.0. You can decrease or increase the number of stages using the arrow buttons for **Number of Stages** in the **Multi-Stage** tab. (If you want to increase the number of stages beyond five, you will need to use the text-interface command `solve/set/multi-stage`.)

For each stage, you can modify the **Coefficient**. Coefficients must be greater than 0 and less than 1. The final stage should always have a coefficient of 1.

28.7.1.2. Controlling Updates to Dissipation and Viscous Stresses

For each stage, you can indicate whether or not artificial dissipation and viscous stresses are evaluated. If a **Dissipation** box is selected for a particular stage, artificial dissipation will be updated on that stage. If not selected, artificial dissipation will remain "frozen" at the value of the previous stage. If a **Viscous** box is selected for a particular stage, viscous stresses will be updated on that stage. If not selected, viscous stresses will remain "frozen" at the value of the previous stage. Viscous stresses should always be computed on the first stage, and successive evaluations will increase the "robustness" of the solution process, but will also increase the expense (that is, increase the CPU time per iteration). For steady problems, the final solution is independent of the stages on which viscous stresses are updated.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.7.1.3. Resetting the Multi-Stage Parameters

If you change the multi-stage parameters, but you then want to return to the default scheme set by ANSYS Fluent, you can click the **Default** button in the **Multi-Stage** tab in the Advanced Solution Controls Dialog Box (p. 2513). ANSYS Fluent will change the values to the defaults and the **Default** button will become the **Reset** button. To get your values back again, you can click the **Reset** button.

28.8. Selecting Gradient Limiters

The default gradient limiter in ANSYS Fluent is the **Standard** limiter. Each of the limiters is described in detail in Gradient Limiters in the Theory Guide. The gradient limiters are accessible from the **Expert** tab in the **Advanced Solution Controls** dialog box.

Solution → **Controls** **Advanced...**

You can select **Standard**, **Multidimensional**, or **Differentiable** from the **Spatial Discretization Limiter Type** drop-down list.

Each of these options can also be accessed by typing the following text command:

`solve` → `set` → `slope-limiter-set`

Choose from the following options:

Criterion	Type
0	Default (TVD) slope limiter
1	Multidimensional (TVD) slope limiter
2	Differentiable slope limiter

Note that the `Default(TVD) slope limiter` in the TUI is equivalent to the **Standard** option in the GUI.

For each of the gradient limiter methods, ANSYS Fluent provides two limiting directions:

- **Cell to Face Limiting** is where the limited value of the reconstruction gradient is determined at cell face centers. This is the default method.
- **Cell to Cell Limiting** is where the limited value of the reconstruction gradient is determined along a scaled line between two adjacent cell centroids. On an orthogonal mesh (or when cell-to-cell direction is parallel to face area direction) this method becomes equivalent to the default cell to face method. For smooth field variation, cell to cell limiting may provide less numerical dissipation on meshes with skewed cells.

ANSYS Fluent also provides the option to apply a limiter filter to the **Standard** and **Differentiable** limiters. The purpose of the limiter filter is to maintain higher-order accuracy for the main flow variables. It suppresses limiter intervention arising from small numerical noise, while maintaining limiter control when there are actual discontinuities or large gradients in the solution (such as at shocks, boundary layers, and so on). The limiter filter can also help in improving the apparent convergence of the solution residual.

To use the limiter filter enable **Apply Limiter Filter**. When the limiter filter is turned on, then by default the filter mechanism is applied to the main flow and turbulent variables only. The filter mechanism is based on comparing local cell changes to average global domain changes of a particular flow variable.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the local changes are very small compared to the average global changes, then the limiter intervention will be suppressed.

Note

The limiter filter is not available when using the **Multidimensional** limiter.

28.9. Initializing the Solution

Before starting your CFD simulation, you must provide ANSYS Fluent with an initial "guess" for the solution flow field. In many cases, you must take extra care to provide an initial solution that will allow the desired final solution to be attained. A real-life supersonic wind tunnel, for example, will not "start" if the back pressure is simply lowered to its operating value; the flow will choke at the tunnel throat and will not transition to supersonic. The same holds true for a numerical simulation: the flow must be initialized to a supersonic flow or it will simply choke and remain subsonic.

There are two methods for initializing the solution:

- Initialize the entire flow field (in all cells). Three methods are available:
  - Standard initialization (see Initializing the Entire Flow Field Using Standard Initialization (p. 1591))
  - FMG initialization (see Full Multigrid (FMG) Initialization (p. 1595))
  - Hybrid initialization (see Hybrid Initialization (p. 1597))
- Patch values or functions for selected flow variables in selected cell zones or "registers" of cells. (Registers are created with the same functions that are used to mark cells for adaption.)

Important

Before patching initial values in selected cells, you must first initialize the entire flow field. You can then patch the new values over the initialized values for selected variables.

For additional information, see the following sections:
28.9.1. Initializing the Entire Flow Field Using Standard Initialization
28.9.2. Patching Values in Selected Cells

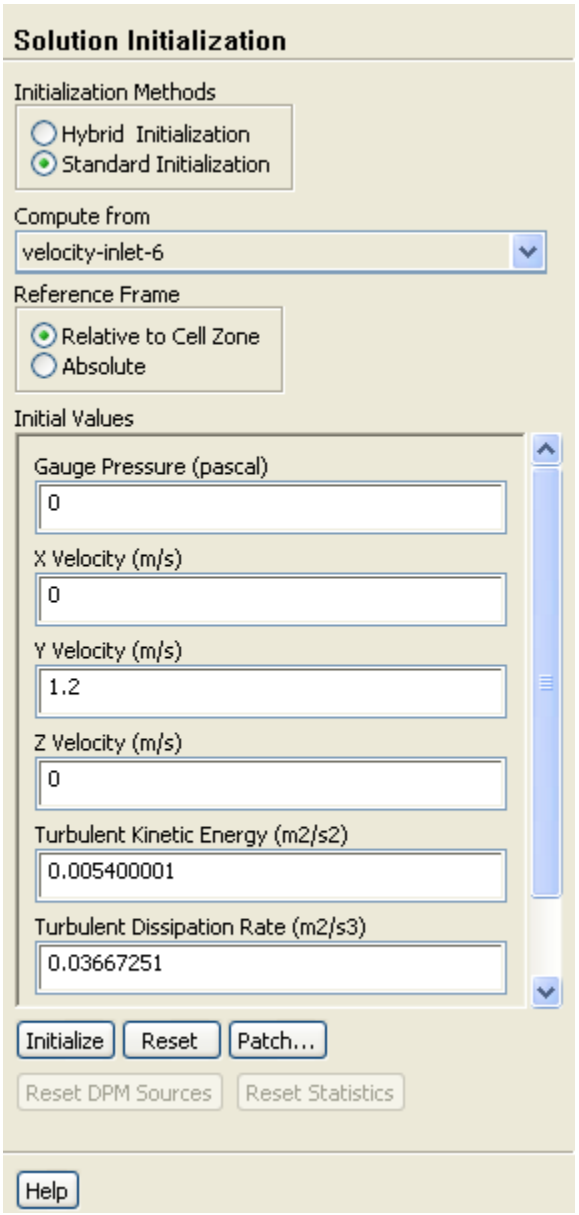
28.9.1. Initializing the Entire Flow Field Using Standard Initialization

Before you start your calculations *or* patch initial values for selected variables in selected cells (Patching Values in Selected Cells (p. 1593)) you must initialize the flow field in the entire domain. The Solution Initialization Task Page (p. 2520) (Figure 28.13: The Solution Initialization Task Page (p. 1592)) allows you to set initial values for the flow variables and initialize the solution using these values.

Solution → **Initialization**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.13: The Solution Initialization Task Page



You can compute the values from information in a specified zone, enter them manually, or have the solver compute average values based on all zones. You can also indicate whether the specified values for velocities are absolute or relative to the velocity in each cell zone. The steps for standard initialization are as follows:

1. Select **Standard Initialization** as the **Initialization Method**.

2. Set the initial values:

   - To initialize the flow field using the values set for a particular zone, select the zone name in the **Compute from** drop-down list. All values under the **Initial Values** heading will automatically be computed and updated based on the conditions defined at the selected zone.

   - To initialize the flow field using computed average values, select **all-zones** in the **Compute from** drop-down list. ANSYS Fluent will compute and update the **Initial Values** based on the conditions defined at all boundary zones.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you want to change one or more of the values, you can enter new values manually in the fields next to the appropriate variables. If you prefer to enter all values manually, you can do so without selecting a zone in the **Compute from** list. For species transport cases with a single phase flow, you can select boundary species to be displayed in the list using the **Select Boundary Species** dialog box (see Figure 14.3: The Select Boundary Species Dialog Box (p. 966)) that can be accessed by clicking **Species** in the **Solution Initialization** task page.

3. If your problem involves moving reference frames or sliding meshes, indicate whether the initial velocities are absolute velocities or velocities relative to the motion of each cell zone by selecting **Absolute** or **Relative to Cell Zone** under **Reference Frame**. (If no zone motion occurs in the problem, the two options are equivalent.) The default reference frame for velocity initialization in ANSYS Fluent is relative. If the solution in most of your domain is rotating, using the relative option may be better than using the absolute option.

4. After you are satisfied with the **Initial Values** displayed in the task page, you can click the **Initialize** button to initialize the flow field. If solution data already exist (that is, if you have already performed some calculations or initialized the solution), you must confirm that it is OK to overwrite those data.

28.9.1.1. Saving and Resetting Initial Values

When you initialize the solution by clicking on **Initialize**, the initial values will also be saved; should you need to reinitialize the solution later, you will find the correct values in the task page when you reopen it.

If you accidentally select the wrong zone from the **Compute from** list or manually set a value incorrectly, you can use the **Reset** button to reset all fields to their "saved" values.

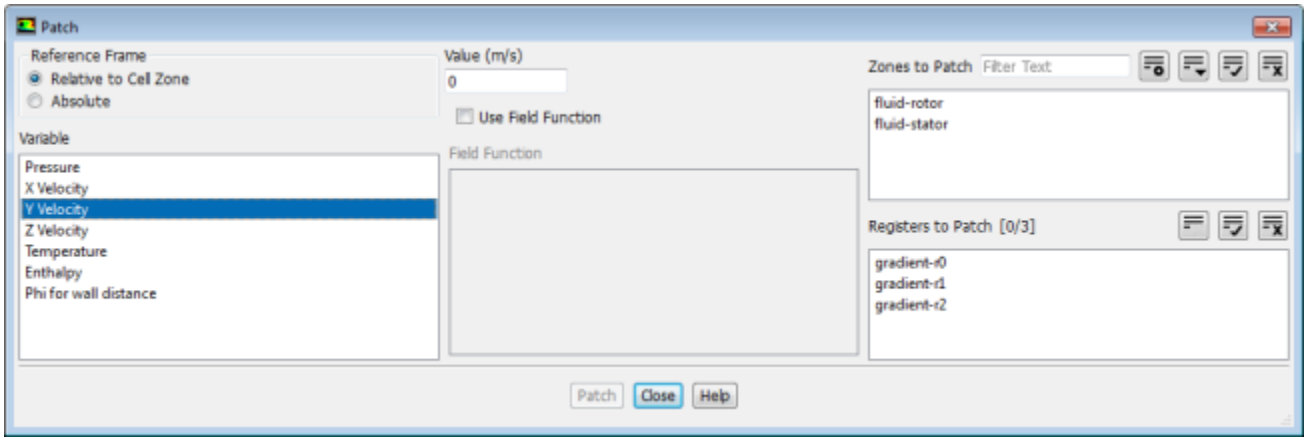
28.9.2. Patching Values in Selected Cells

Once you have initialized (or calculated) the entire flow field, you may patch different values for particular variables into different cells. If you have multiple fluid zones, for example, you may want to patch a different temperature in each one. You can also choose to patch a custom field function (defined using the Custom Field Function Calculator Dialog Box (p. 2683)) instead of a constant value. If you are patching velocities, you can indicate whether the specified values are absolute velocities or velocities relative to the cell zone's velocity. All patching operations are performed with the Patch Dialog Box (p. 2522) (Figure 28.14: The Patch Dialog Box (p. 1594)).

Solution → **Initialization** **Patch...**

Solution → **Initialization** → **Patch...**

Figure 28.14: The Patch Dialog Box



1. Select the variable to be patched in the **Variable** list.

   Note

   For species transport cases with a single phase flow, you can select boundary species to be displayed in the **Variable** list using the **Select Boundary Species** dialog box (see Figure 14.3: The Select Boundary Species Dialog Box (p. 966)) that can be accessed by clicking the **Species** button in the **Solution Initialization** task page.

2. In the **Zones to Patch** and/or **Registers to Patch** lists, choose the zone(s) and/or register(s) for which you want to patch a value for the selected variable.

   Important

   When a shell zone has been created (that is, by enabling shell conduction for a wall and running the calculation), the name of the shell zone will be listed in the **Zones to Patch** list as **shell :** <*wall-name*>, where <*wall-name*> is the name of the wall in which shell conduction has been enabled. Only temperature can be patched into the cells of a shell, and the same value or field function will be patched into every layer of the shell.

3. If you want to patch a constant value, simply enter that value in the **Value** field. If you want to patch a previously-defined field function, enable the **Use Field Function** option and select the appropriate function in the **Field Function** list.

4. If you selected a velocity in the **Variable** list, and your problem involves moving reference frames or sliding meshes, indicate whether the patched velocities are absolute velocities or velocities relative to the motion of each cell zone by selecting **Absolute** or **Relative to Cell Zone** under **Reference Frame**. (If no zone motion occurs in the problem, the two options are equivalent.) The default reference frame for velocity patching in ANSYS Fluent is relative. If the solution in most of your domain is rotating, using the relative option may be better than using the absolute option.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click the **Patch** button to update the flow-field data. (Note that patching will have no effect on the iteration or time-step count.)

Important

If you apply a patch when setting up an ANSYS Fluent simulation from ANSYS Workbench, the patch will not be automatically performed during future automatic solution updates from Workbench. If you need to apply the patch each time the solution is updated from Workbench, you can do so by adding the text command `/solve/patch` (with appropriate arguments) to the **Original Settings** command list in the **Case Modification** tab of the **Automatic Solution Initialization and Case Modification** dialog box. See Automatic Initialization of the Solution and Case Modification (p. 1642) in this manual and Case Modification Strategies with Fluent and Workbench in the Fluent in Workbench User's Guide for more details on automatic case modification.

28.9.2.1. Using Registers

The ability to patch values in cell registers gives you the flexibility to patch different values within a single cell zone. For example, you may want to patch a certain value for temperature only in fluid cells with a particular range of concentrations for one species. You can create a cell register (basically a list of cells) using the functions that are used to mark cells for adaption. These functions allow you to mark cells based on physical location, cell volume, gradient or isovalue of a particular variable, and other parameters. See Adapting the Mesh (p. 1681) for information about marking cells for adaption. Manipulating Adaption Registers (p. 1701) provides information about manipulating different registers to create new ones. Once you have created a register, you can patch values in it as described above.

28.9.2.2. Using Field Functions

By defining your own field function using the Custom Field Function Calculator Dialog Box (p. 2683), you can patch a non-constant value in selected cells. For example, you may want to patch varying species mass fractions throughout a fluid region. To use this feature, simply create the function as described in Custom Field Functions (p. 2003), and then perform the function-patching operation in the Patch Dialog Box (p. 2522), as described above.

28.9.2.3. Using Patching Later in the Solution Process

Since patching affects only the variables for which you choose to change the value, leaving the rest of the flow field intact, you can use it later in the solution process without losing calculated data. (Initialization, on the other hand, resets all data to the initial values). For example, you might want to start a combustion calculation from a cold-flow solution. You can simply read in (or calculate) the cold-flow data, patch a high temperature in the appropriate cells, and continue the calculation.

Patching can also be useful when you are solving a problem using a step-by-step technique, as described in Step-by-Step Solution Processes (p. 1669).

28.10. Full Multigrid (FMG) Initialization

For many complex flow problems such as those found in rotating machinery, or flows in expanding or spiral ducts, flow convergence can be accelerated if a better initial solution is used at the start of the calculation. The Full Multigrid initialization (FMG initialization) can provide this initial and approximate solution at a minimum cost to the overall computational expense.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For more information about FMG initialization, see Overview of FMG Initialization in the Theory Guide.

For additional information, see the following sections:
28.10.1. Steps in Using FMG Initialization
28.10.2. Convergence Strategies for FMG Initialization

28.10.1. Steps in Using FMG Initialization

You can access the FMG initialization procedure using the text user interface (TUI) once the standard flow initialization is performed (see Full-Approximation Storage (FAS) Multigrid in the Theory Guide) or if valid flow data is available (that is, through reading a data file).

To customize the FMG initialization, type the following command :

`solve` → `initialize` → `set-fmg-initialization`

You will be asked to enter:

- The number of multigrid levels for the FMG iteration (the default is 5).

  Important

  For small cases (100, 000 cells or less), it is recommended that you lower the number of multigrid levels to 3 or 4.

- For each level of multigrid, you will be asked to enter the residual reduction (the default value is `0.001`), and the number of cycles per level (the defaults at each level are `10`, `10`, `50`, `100`, `500`, and `500`). In general, you should perform more iterations on coarse levels than fine levels. Level 0 is the finest level, which represents the original mesh.
- FMG iteration Courant-number (the default is `0.75`). This will be the CFL value that the FAS multigrid will use for the FMG initialization.
- Enabling verbose mode (the default is `no`). By enabling this option, you will be able to monitor the convergence at each level.

  Important

  If you do not customize the FMG settings, then the default values will be used.

To perform the FMG initialization, type the following command :

`solve` → `initialize` → `fmg-initialization`

When you are prompted to `Enable FMG initialization? [no]`, type `yes`.

When verbose mode is selected and the FMG initialization is being used, ANSYS Fluent will first output the multigrid level information followed by convergence history for the FAS multigrid cycle on each level. The normalized residual value is printed after ten FAS cycles or when the number of FAS cycles is reached. The output will indicate when convergence is reached on each level and when the solution is being interpolated to the next level.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.10.2. Convergence Strategies for FMG Initialization

When setting the FMG initialization parameters, you should consider performing more iterations on the coarse levels than on the fine levels. However, remember that the purpose of FMG initialization is to obtain a good initial solution at a low cost. You should try to avoid unreasonable convergence tolerance that will make the FMG initialization expensive.

Turn on the verbose mode to help you determine if the flow is converging as expected during the FMG iterations. If the solution is not converging to the desired tolerance, consider increasing the number of FAS multigrid cycles at each level. If the solution is diverging during the FAS cycles, then consider lowering the FMG iteration Courant number since the default value is probably too aggressive and is likely causing the solution to diverge.

For turbulent flows, it is very important to first perform standard initialization with proper and realistic values of the turbulence variables (for example k and ε). This can be done by computing the average values based on the conditions defined at the inflow boundary or at all boundary zones. Then, you can proceed with FMG initialization. Unrealistic initialization of turbulence variables may cause convergence difficulties during the first few iterations on the fine mesh, thereby nullifying the benefit of FMG initialization.

28.11. Hybrid Initialization

Hybrid initialization is yet another initialization method in ANSYS Fluent. The other initialization methods are standard initialization and FMG initialization. Hybrid initialization is a collection of recipes and boundary interpolation methods. It solves Laplace's equation to determine the velocity and pressure fields. All other variables, such as temperature, turbulence, species fractions, volume fractions, and so on, will be automatically patched based on domain averaged values or a particular interpolation recipe.

For more information about hybrid initialization, see Hybrid Initialization in the Theory Guide.

For additional information, see the following sections:
28.11.1. Steps in Using Hybrid Initialization
28.11.2. Solution Strategies for Hybrid Initialization

28.11.1. Steps in Using Hybrid Initialization

The default initialization method for single phase steady-state flows is the **Hybrid Initialization** method.

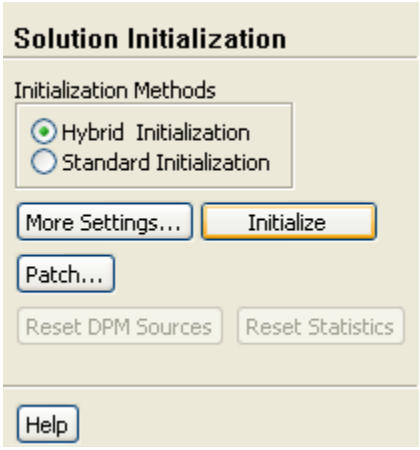
Note

For other flow types, such as multiphase or unsteady simulations, the default initialization method is the **Standard Initialization** method. However both initialization methods are available for use in all flow conditions and types.

To use **Hybrid Initialization**, go to the Solution Initialization Task Page (p. 2520) (Figure 28.15: The Solution Initialization Task Page for Hybrid Initialization (p. 1598)) where you will select **Hybrid Initialization**.

Solution → **Initialization**

Figure 28.15: The Solution Initialization Task Page for Hybrid Initialization

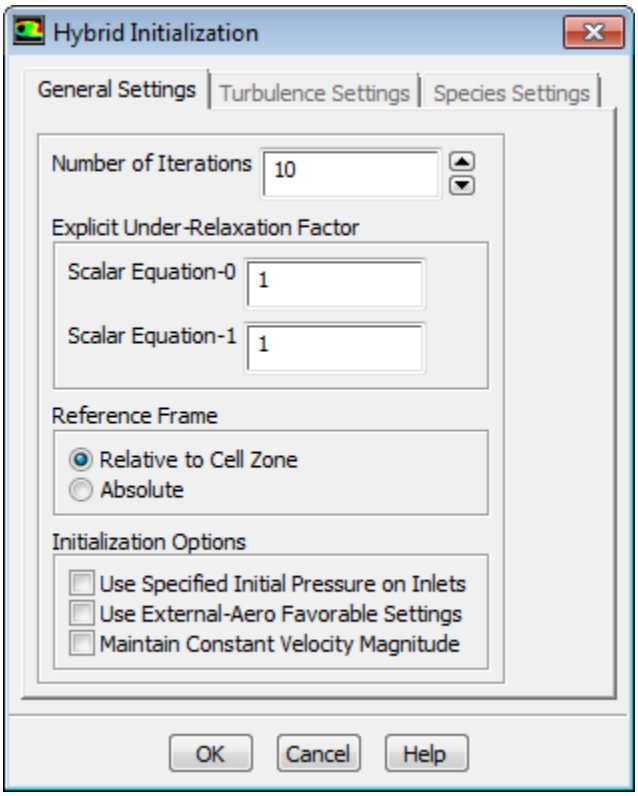


Note

In most cases, you need not do anything more than click the **Initialize** button. However, should you decide to modify the default settings for the hybrid initialization method, click **More Settings...**.

If you click **More Settings...**, the **Hybrid Initialization** dialog box (Figure 28.16: The Hybrid Initialization Dialog Box (p. 1598)) will open. A host of settings that control the **Hybrid Initialization** strategy will be available for you to adjust.

Figure 28.16: The Hybrid Initialization Dialog Box



You can make adjustments in three different areas:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **General Settings** tab:

  – **Number of Iterations** uses a default value of 10. This is the number of iterations that will be performed while solving the Laplace equations to initialize the velocity and pressure. In general, you do not need to change the number of iterations. However, for complex and highly curved geometries, if the default number of iterations is not enough to reach the convergence tolerance of 1e-06 and the flow fields are not to your liking, then you may want to increase the number of iterations and re-initialize the flow.

  – **Explicit Under-Relaxation Factor** uses a default value of 1. This value will be used while solving the Laplace equation to initialize the velocity and pressure. In general, you do not need to change the explicit under-relaxation factor. However, for some cases, where the scalar residuals are oscillating and showing difficulty reaching the convergence tolerance of 1e-06, you may want to re-initialize the flow by reducing the under-relaxation factor. You may also want to increase the number of iterations to produce a smooth initialization field for the velocity and pressure.

  – **Reference Frame** is set to **Relative to Cell Zone** by default. If your problem involves moving reference frames or sliding meshes, indicate whether the initial velocities are absolute velocities or velocities relative to the motion of each cell zone by selecting **Absolute** or **Relative to Cell Zone**. If no zone motion occurs in the problem, the two options are equivalent. If the solution in most of your domain is rotating, using the relative option may be better than using the absolute option.

  – **Initialization Options** allows you to include the following options:

    → **Use Specified Initial Pressure on Inlets** if you want the specified pressure for **Supersonic/Initialization Gauge Pressure** at the inlet boundaries to be used for solving the Laplace equation for the pressure. Otherwise, ANSYS Fluent uses a predetermined recipe to determine the initial pressure field, as described in Hybrid Initialization of the Theory Guide.

    → **Use External-Aero Favorable Settings** if you want to have the velocity potential patched with a linear value to help accelerate convergence of **Scalar Equation–0** and to obtain a better guess of the velocity field for external-aero problems, such as flow over wings, airfoils, or automobiles.

    → **Maintain Constant Velocity Magnitude** if you want to use the flow direction obtained from solving the velocity potential (**Scalar Equation–0**), while maintaining a constant velocity magnitude throughout the computational domain. This option is helpful in some incompressible external flow problems, porous media problems, or if there are narrow channels where large undesirable velocities can be reached.

- **Turbulence Settings** tab uses by default the domain averaged values for the turbulence parameters. If you want to use variable turbulence parameters you can deselect the **Average Turbulent Parameters** check box. When this option is disabled, then it calculates the turbulent parameters, such as kinetic energy, dissipation energy, and so on, using local flow parameters.

- **Species Settings** tab will by default initialize secondary species with zero mass or mole fractions. If you want to specify the appropriate value for the species, you will need to enable **Specify Species Parameters**. Note that only the boundary species that you have selected in the **Select Boundary Species** dialog box appear in the list (see Figure 14.3: The Select Boundary Species Dialog Box (p. 966)). You can add or remove a species using the **Select Boundary Species** that can be accessed by clicking **Species** in the **Solution Initialization** task page.

28.11.2. Solution Strategies for Hybrid Initialization

In general, you do not need to make any extra adjustments to the hybrid initialization default settings. However, if the hybrid initialization is not producing the initial field to your liking, then you can play with the various options available in the **Hybrid Initialization** dialog box (described in Steps in Using

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Hybrid Initialization (p. 1597)), or you can also use the patching option in addition to the **Hybrid Initialization**. For example, if you are solving a user-defined scalar, then hybrid initialization will initialize them with a value of zero. However, you can specify the value with which you want to patch.

Note

You can also use a user-defined function to initialize the flow or certain flow variable in conjunction with hybrid initialization.

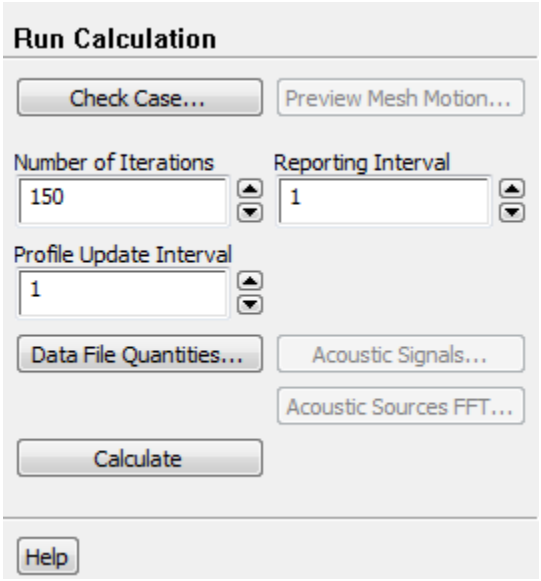
Standard Initialization is the recommended initialization method for porous media simulations. The default **Hybrid initialization** method does not account for the porous media properties, and depending on boundary conditions, may produce an unrealistic initial velocity field. For porous media simulations, the **Hybrid initialization** method can only be used with the **Maintain Constant Velocity Magnitude** option.

28.12. Performing Steady-State Calculations

For steady-state calculations, you will request the start of the solution process using the Run Calculation Task Page (p. 2537) (Figure 28.17: The Run Calculation Task Page (p. 1600)).

Solution → Run Calculation

Figure 28.17: The Run Calculation Task Page



Here, you will supply the number of additional iterations to be performed in the **Number of Iterations** field. (For unsteady calculation inputs, see User Inputs for Time-Dependent Problems (p. 1609)). If no calculations have been performed yet, ANSYS Fluent will begin calculations starting at iteration 1, using the initial solution. If you are starting from current solution data, ANSYS Fluent will begin at the last iteration performed, using the current solution data as its starting point.

By default, ANSYS Fluent will update the convergence monitors (described in Monitoring Solution Convergence (p. 1624)) after each iteration. If you increase the **Reporting Interval** from the default of 1 you can get reports less frequently. For example, if you set the **Reporting Interval** to 2, the monitors will print or plot reports at every other iteration. Note that the **Reporting Interval** also specifies how often ANSYS Fluent should check if the solution is converged. For example, if your solution converges

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

after 40 iterations, but your **Reporting Interval** is set to 50, ANSYS Fluent will continue the calculation for an extra 10 iterations before checking for (and finding) convergence.

When you click the **Calculate** button, ANSYS Fluent will begin to calculate. During iteration, a **Working** dialog box is displayed. Clicking the **Cancel** button or typing **Ctrl**+**c** in the ANSYS Fluent console will stop after the current iteration (in steady-state simulations) or the current time step (in transient simulations). If you are running a transient simulation and you want to interrupt the solution before the end of the current time step you can type **Ctrl**+**c** a second time. (See below for more details).

For additional information, see the following sections:

28.12.1. Updating UDF Profiles
28.12.2. Interrupting Iterations
28.12.3. Resetting Data

28.12.1. Updating UDF Profiles

If you have used a user-defined function (UDF) to define any boundary conditions you can control the frequency with which the function is updated by modifying the value of the **Profile Update Interval**. If **Profile Update Interval** is set to n, the function will be updated after every n iterations.

By default, the **Profile Update Interval** is set to 1. You might want to increase this value if your profile computation is expensive. See the Fluent Customization Manual for details about creating and using UDFs.

28.12.2. Interrupting Iterations

As mentioned above, you can interrupt the calculation by clicking the **Cancel** button in the **Working** dialog box that appears while the solver is calculating or by typing **Ctrl**+**c** in the console. This will stop the calculation after the current iteration (in steady-state) or after the current time step (in transient). This allows you to stop the calculation process and then start again smoothly with additional iterations or time steps. If you are running a transient simulation and want to interrupt the calculation without waiting for the end of a time step you can type **Ctrl**+**c** a second time.

28.12.3. Resetting Data

After you have performed some iterations, if you decide to start over again from the first iteration (for example, after making some changes to the problem setup), you can reinitialize the solution using the Solution Initialization Task Page (p. 2520), as described in Initializing the Entire Flow Field Using Standard Initialization (p. 1591).

28.13. Performing Pseudo Transient Calculations

For steady-state calculations, when using the pressure-based coupled solver or the density-based implicit solver, you have the option of solving your flow in a pseudo-transient fashion. The pseudo transient under-relaxation method is a form of implicit under-relaxation, described in Pseudo Transient Under-Relaxation in the Theory Guide.

To apply the pseudo transient under-relaxation method, perform the following:

1. Select the **Density-Based** solver or the **Pressure-Based** solver.

   Setup → **General**

2. Go to the **Solution Methods** task page (Figure 28.18: The Solution Methods Task Page (p. 1602)).

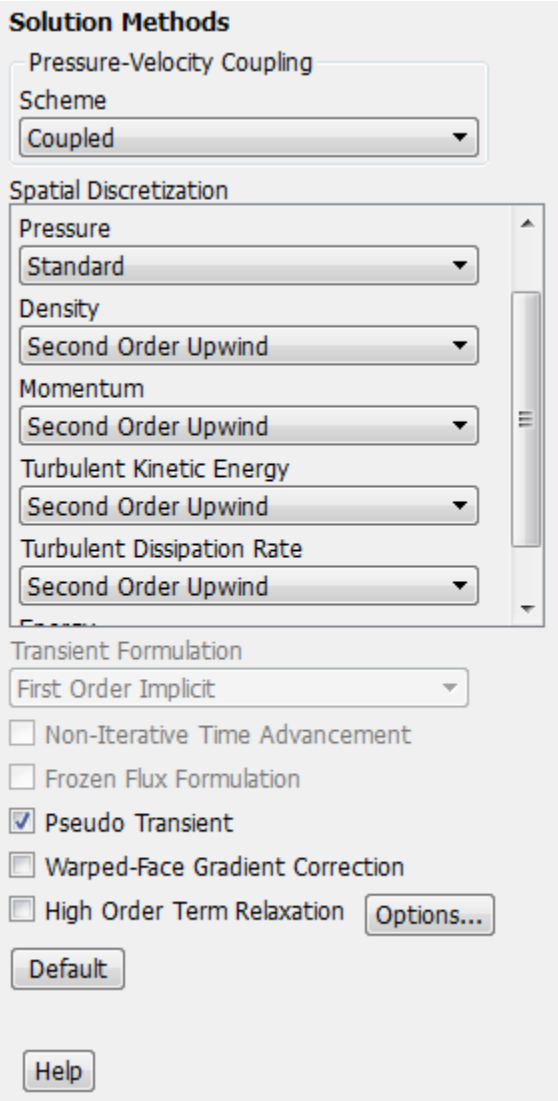
   Solution → **Methods**

   a. If you are using the pressure-based solver, choose the **Coupled** scheme under **Pressure-Velocity Coupling**.

   b. If you are using the density-based solver, choose the **Implicit** scheme under **Formulation**.

   c. Enable the **Pseudo Transient** option.

Figure 28.18: The Solution Methods Task Page



28.13.1. Setting Pseudo Transient Explicit Relaxation Factors

In addition to pseudo transient under-relaxation, you can specify an explicit under-relaxation of the equation to control the update of computed variables at each iteration (see Pseudo Transient Under-Relaxation in the Theory Guide). The default values of under-relaxation parameters for all variables are set to values that work well for most of the cases. It is good practice to start a calculation with the default

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

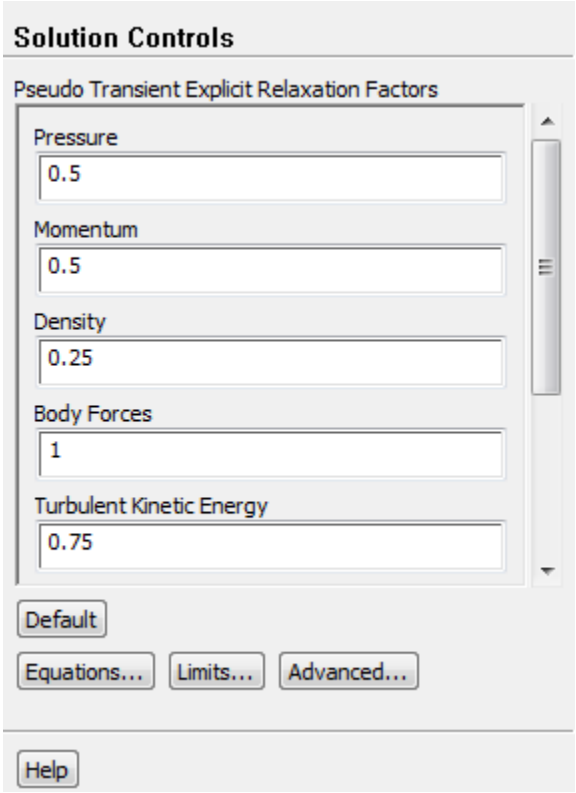
under-relaxation parameters. If your case exhibits divergence or the residuals continue to increase after a few iterations, then you should reduce the under-relaxation factors.

28.13.1.1. User Inputs

You can modify the pseudo transient under-relaxation factors in the Solution Controls Task Page (p. 2508) (Figure 28.19: The Solution Controls Task Page for the Pseudo Transient Runs (p. 1603)).

Solution → Controls

Figure 28.19: The Solution Controls Task Page for the Pseudo Transient Runs



You can set the under-relaxation factor for each equation in the field next to its name under **Pseudo Transient Explicit Relaxation Factors**.

Important

If you are using the pressure-based solver, all equations will have an associated under-relaxation factor (see Under-Relaxation of Equations in the Theory Guide). If you are using the density-based solver, only those equations that are solved sequentially (see Density-Based Solver in the Theory Guide) will have under-relaxation factors.

If you change under-relaxation factors, but you then want to return to ANSYS Fluent's default settings, you can click the **Default** button.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.13.2. Setting Solution Controls for the Pseudo Transient Method

To have further control over the parameters for each individual equation, when solving a pseudo transient case, you can go to the **Expert** tab, in the **Advanced Solution Controls** dialog box (Figure 28.20: The Advanced Solution Controls Dialog Box for the Pseudo Transient Method (p. 1605)). Note that all equations, except for flow equations (that is pressure and momentum) will be listed. Generally, you will not need to visit this dialog box to enter equation-specific solution parameters. However, it may help in cases where a particular equation is giving convergence problems. Here, ANSYS Fluent allows two options to improve convergence:

1. Specify a time scale factor for the equation specific time step in lieu of using a uniform global pseudo time step. This scaling factor scales the pseudo time step employed for the flow equations specified in the **Run Calculation** task page (see Solving Pseudo-Transient Flow (p. 1605)).

2. Use the standard steady-state method by turning off pseudo transient for that particular equation. Here, the corresponding under-relaxation factor to be employed with that equation may be specified. The default values of under-relaxation parameters for all variables are set to values that work well for most of the cases.

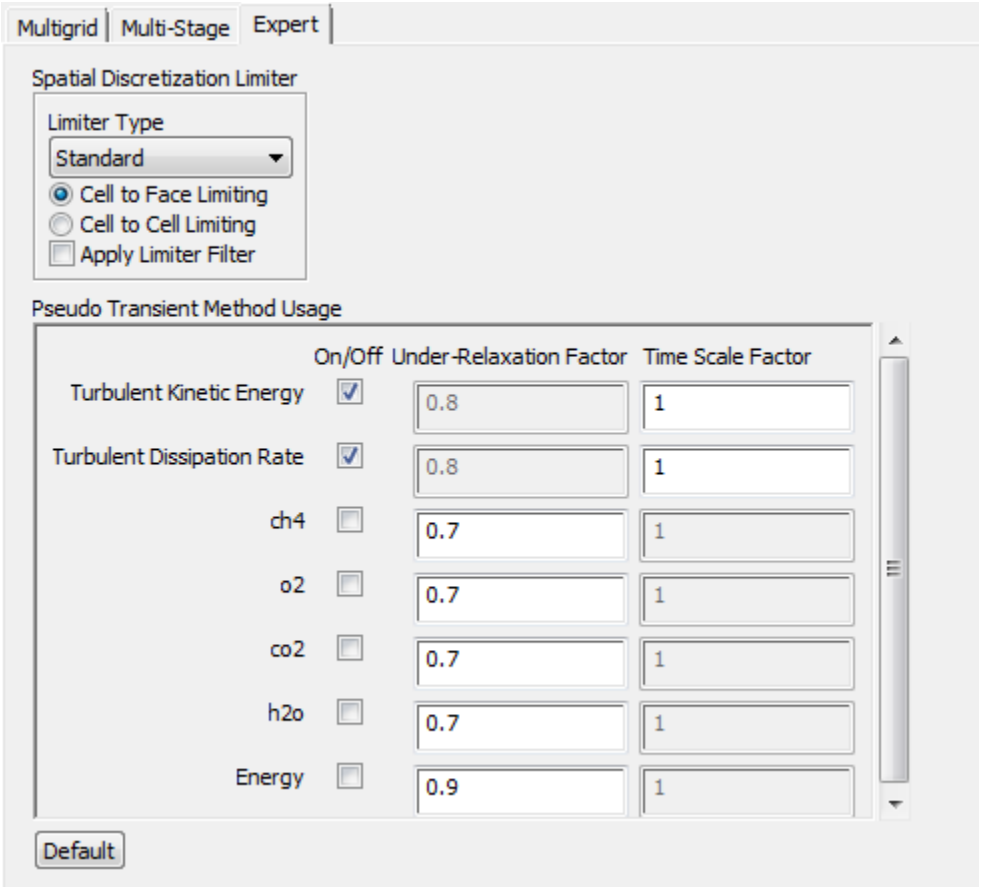
The default setting will have the pseudo transient method turned on for all equations with the corresponding time scale factor set to unity, except when one of the combustion models is used. When the pseudo transient method is enabled In combustion cases the species, enthalpy, and combustion variable equations will only use the pseudo transient method if it is manually enabled in the **Expert** tab. As well, when using the premixed, partially-premixed, or PDF combustion models, the energy equation will only use the pseudo transient method if it is manually enabled in the **Expert** tab.

By default, all user-defined scalars (UDS) have pseudo transient method disabled. This is because the physical parameters of the UDS and the appropriate time scale are not known to the solver before starting the calculation. You can manually enable pseudo transient method for UDS under the **Expert** tab in the Advanced Solution Controls Dialog Box (p. 2513).

Solution → Controls Advanced...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.20: The Advanced Solution Controls Dialog Box for the Pseudo Transient Method



Specify the equation-specific steady-state solution method for a particular equation (if needed) by enabling or disabling the pseudo transient method using the **On/Off** check box next to the equation. The dialog box then allows either specification of a pseudo **Time Scale Factor** or **Under-Relaxation Factor** for that particular equation based on the check box setting.

Note

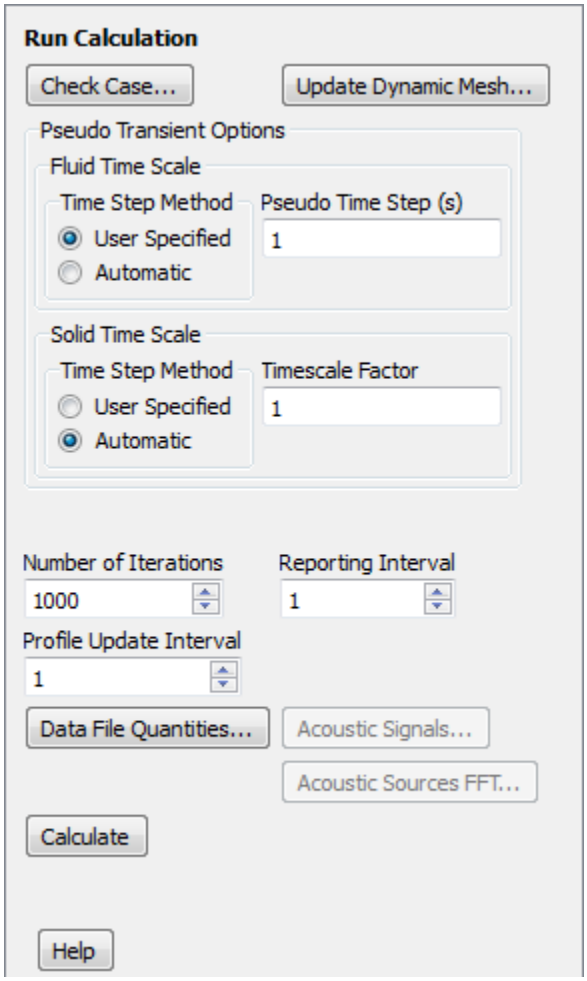
For multiphase flows, the pseudo transient expert options for the volume fraction equation are available only when it is solved in segregated fashion. It is not available when the **Coupled with Volume Fractions** option is enabled in the **Solution Methods** task page (see Selecting the Pressure-Velocity Coupling Method (p. 1507) for information about this setting).

28.13.3. Solving Pseudo-Transient Flow

With the **Pseudo Transient** option enabled in the **Solution Methods** task page, you can now specify the time step for the **Fluid Zone** and/or the **Solid Zone** under **Pseudo Transient Options** in the **Run Calculation** task page (Figure 28.21: The Run Calculation Task Page for the User Specified Pseudo Transient Option (p. 1606)).

Solution → Run Calculation

Figure 28.21: The Run Calculation Task Page for the User Specified Pseudo Transient Option



1. Select the **Time Step Method** for the **Fluid Time Scale**.

   a. If you choose **User Specified**, you will enter the **Pseudo Time Step**, which is used for every equation unless the equation specific time step is used for a particular equation, as mentioned in Setting Solution Controls for the Pseudo Transient Method (p. 1604).

   b. If you choose **Automatic**, which is the default method, (see Figure 28.22: The Run Calculation Task Page for the Automatic Pseudo Transient Option (p. 1607)), the pseudo time step is calculated internally and used for each equation listed, unless the equation has a time step specified as noted in Setting Solution Controls for the Pseudo Transient Method (p. 1604). For more information about the automatic time step calculation, refer to Automatic Pseudo Transient Time Step in the Theory Guide.

      Select the Length Scale Method. Three options are available in the drop-down list:

      i. **Conservative** is the default length scale calculation method, for which you will enter the **Timescale Factor**. The **Timescale Factor** is a scaling factor used to scale the calculated time step. The default value is 1.0. You can increase or decrease it to increase or decrease the size of the time step, respectively.

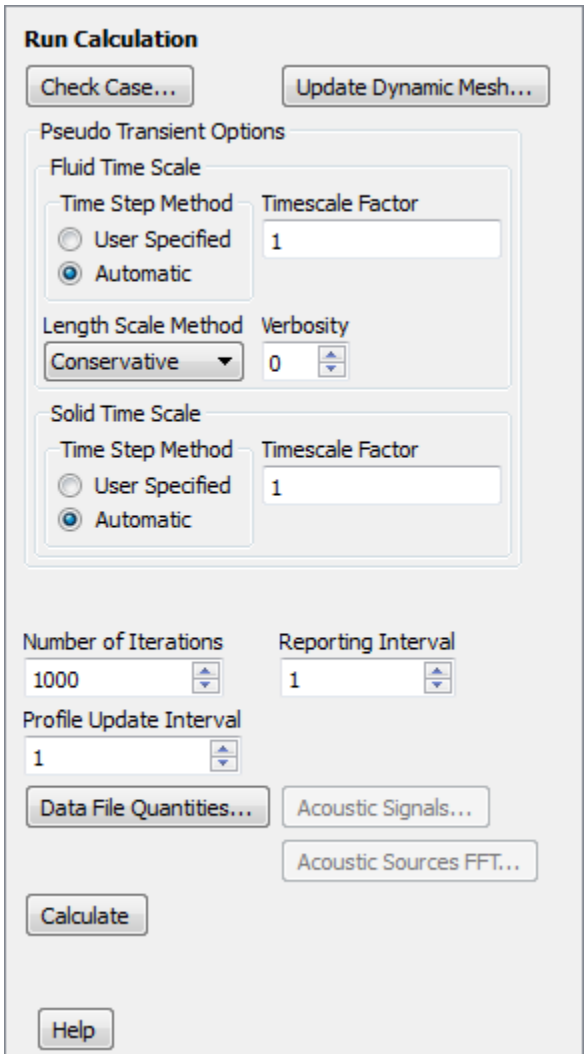
      ii. **Aggressive** predicts a higher time step size than the **Conservative** method. You will also need to specify the **Timescale Factor**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

iii. **User Specified** allows you to control the input for the **Length Scale**. This method is particularly useful in problems that have a specific length scale that is difficult to determine from the overall geometry of the problem. For example, in the case of flow over an airfoil, the appropriate length scale would be the length of the airfoil instead of the length scale calculated based on the geometry of the entire domain.

iv. You can specify the **Verbosity**. This is an integer value of 0 or 1. The default value is 0. If you want to print the pseudo time step size, enter a value of 1 for the **Verbosity**.

Figure 28.22: The Run Calculation Task Page for the Automatic Pseudo Transient Option



2. Select the **Time Step Method** for the **Solid Time Scale**.

Note

Solid Time Scale only appears in the interface when a solid zone is present in the domain or when the **Energy** equation is enabled along with **Porous Zone** or the **Solidification & Melting** model.

---

a. If you choose **User Specified**, you will enter the **Pseudo Time Step**, which is used where the **Solid Time Scale** is needed (that is, solid zones, porous zones, and/or solidification/melting model).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. If you choose **Automatic**, which is the default method, the pseudo time step is calculated internally as described in Automatic Pseudo Transient Time Step in the Theory Guide. Specify a **Timescale Factor** to control/adjust the pseudo time step obtained from the automatically calculated **Solid Time Scale**.

3. Continue setting up your solution as you would a steady-state run, as described in Performing Steady-State Calculations (p. 1600).

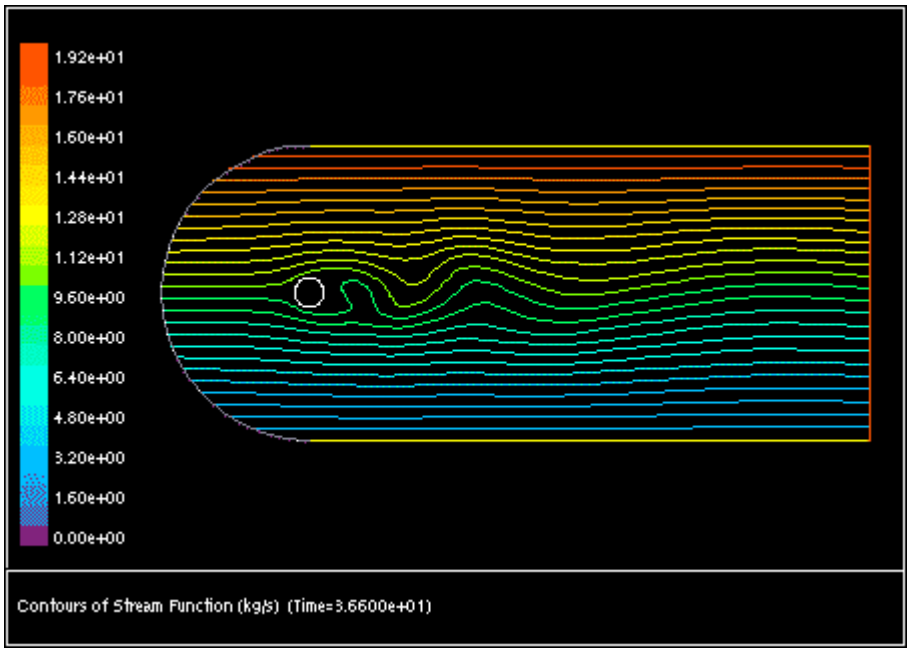
28.14. Performing Time-Dependent Calculations

ANSYS Fluent can solve the conservation equations in a time-dependent manner, to simulate a wide variety of time-dependent phenomena, such as

- vortex shedding and other time-periodic phenomena
- compressible filling and emptying problems
- transient heat conduction
- transient chemical mixing and reactions

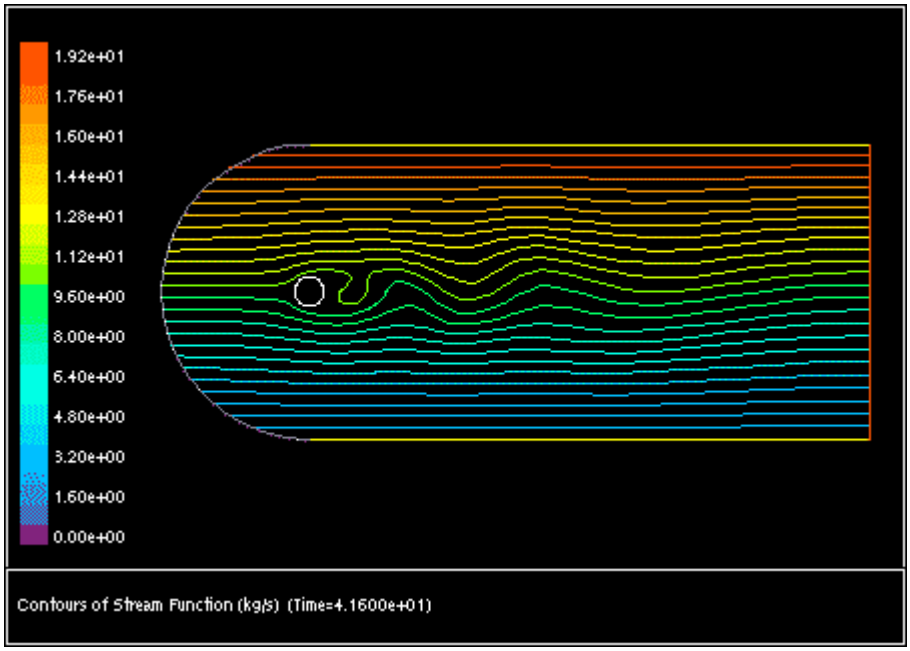
Figure 28.23: Time-Dependent Calculation of Vortex Shedding (t=36.6 sec) (p. 1608) and Figure 28.24: Time-Dependent Calculation of Vortex Shedding (t=41.6 sec) (p. 1609) illustrate the time-dependent vortex shedding flow pattern in the wake of a cylinder.

Figure 28.23: Time-Dependent Calculation of Vortex Shedding (t=36.6 sec)



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.24: Time-Dependent Calculation of Vortex Shedding (t=41.6 sec)



Enabling time dependence is sometimes useful when attempting to solve steady-state problems that tend toward instability (for example, natural convection problems in which the Rayleigh number is close to the transition region). It is possible in many cases to reach a steady-state solution by integrating the time-dependent equations.

For details about temporal discretization, see Temporal Discretization in the Theory Guide.

For additional information, see the following sections:

28.14.1. User Inputs for Time-Dependent Problems
28.14.2. Adaptive Time Stepping
28.14.3. Variable Time Stepping
28.14.4. Postprocessing for Time-Dependent Problems

28.14.1. User Inputs for Time-Dependent Problems

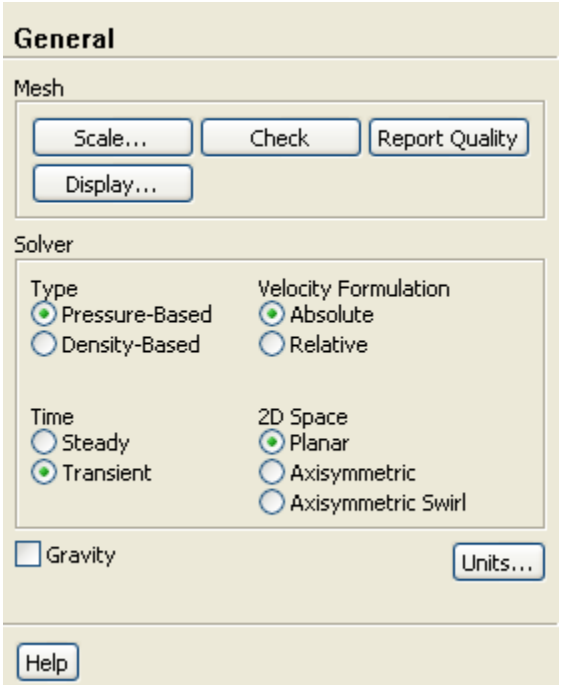
To solve a transient problem, you will follow the procedure outlined below:

1. Enable the **Transient** option in the **General** task page (Figure 28.25: The General Task Page for a Transient Calculation (p. 1610)).

   Setup → **General**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.25: The General Task Page for a Transient Calculation



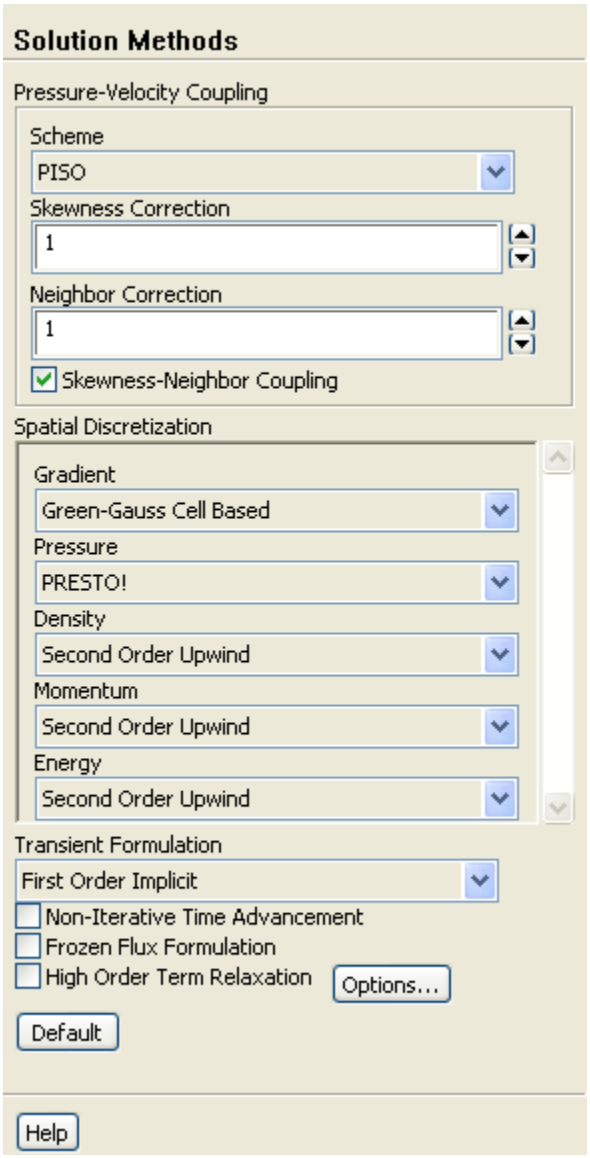
2. Define all relevant models and boundary conditions. Note that any boundary conditions specified using user-defined functions can be made to vary in time. See the Fluent Customization Manual for details.

3. Specify the desired parameters in the **Solution Methods** task page (Figure 28.26: The Solution Methods Task Page for a Transient Calculation (p. 1611)).

   Solution → **Methods**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.26: The Solution Methods Task Page for a Transient Calculation



If you are using the pressure-based solver, select **PISO** from the **Scheme** drop-down list in the **Pressure-Velocity Coupling** group box. To increase the speed of the calculations, you may need to modify the parameters related to the PISO scheme from their default values. See PISO (p. 1562) for more information about the optimal use of the PISO algorithm.

Important

If you are using the LES turbulence model with small time steps, the PISO scheme may be too computationally expensive. It is therefore recommended that you select **SIMPLE** or **SIMPLEC** instead of **PISO**.

Important

It is best to select the **Coupled** pressure-velocity coupling scheme if you are using large time steps to solve your transient flow, or if you have a poor quality mesh.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Next, specify the desired **Transient Formulation**. The **First Order Implicit** formulation is sufficient for most problems. If you need improved accuracy, you can either use **Second Order Implicit** or **Bounded Second Order Implicit**. The **Bounded Second Order Implicit** formulation would provide better stability, since time discretization would always ensure the bounds for variables, if available.

Important

Note that while the **Bounded Second Order Implicit** formulation provides the same accuracy as the **Second Order Implicit** formulation, it actually provides better stability.

Important

The **Bounded Second Order Implicit** formulation is available only for the pressure-based solver, and not for the density-based solver.

The **Explicit** formulation (available only for the density-based solver) is used primarily to capture the transient behavior of moving waves, such as shocks. For details, see Temporal Discretization in the Theory Guide.

When using the pressure-based solver, you have the additional options of selecting **Non-Iterative Time Advancement** and **Frozen Flux Formulation** for your time-dependent flow calculations (see Time-Advancement Algorithm and Steady-State Iterative Algorithm in the Theory Guide, respectively). Note that the latter option is only available for single-phase transient problems that do not use a moving/deforming mesh model.

4. (optional) If you are using the explicit transient formulation with specified Courant number *or* if you are using the adaptive time stepping method (described in a later step and in Adaptive Time Stepping (p. 1618)) it is recommended that you enable the printing of the current time (for the explicit transient formulation) or the current time step size (for the adaptive time stepping method) at each iteration, using Figure 28.34: Report File for 'flow-time', 'delta-time', and 'iters-per-timestep' (p. 1634).

   Solution → **Monitors** → **Statistic** **Edit...**

   Make sure that the desired item is selected from the **Statistics** selection list (**time** for the current time or **delta_time** for the current time step size) and enable the **Print** option. When ANSYS Fluent prints the residuals to the console at each iteration, it will include a column with the current time or the current time step size.

5. (optional) Use the Drag Report Definition Dialog Box (p. 2691), the Lift Report Definition Dialog Box (p. 2724), the Moment Report Definition Dialog Box (p. 2746), or the Surface Report Definition Dialog Box (p. 2807) to monitor (and/or save to a file) time-varying force coefficient values or a report of a field variable or function on a surface as it changes with time. See Monitoring Solution Convergence (p. 1624) for details.

6. Set the initial conditions (at time $t=0$) using the **Solution Initialization** task page.

   Solution → **Initialization**

   You can also read in a steady-state data file to set the initial conditions.

   File → **Read** → **Data...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Use the **Autosave** dialog box to specify the file name and frequency with which case and data files should be saved during the solution process. To open the **Autosave** dialog box, click the **Edit...** button next to **Autosave Every** in the **Calculation Activities** task page.

   Solution → Calculation Activities → Autosave → On

   See Automatic Saving of Case and Data Files (p. 48) for details about automatic file saving.

   The **Calculation Activities** task page also allows you to export solution and particle history data during the transient calculation. See Exporting Data During a Transient Calculation (p. 86) for details.

   If you want to create a graphical animation of the solution over time, you can use Figure 28.43: The Animation Definition Dialog Box (p. 1648) to set up the graphical displays that you want to use in the animation. See Animating the Solution (p. 1647) for details.

   You may also want to request automatic execution of other commands using the Execute Commands Dialog Box (p. 2535). See Executing Commands During the Calculation (p. 1638) for details.

8. (optional) You can improve the convergence of the transient calculations by enabling the **Extrapolate Variables** option in the Run Calculation Task Page (p. 2537). This option instructs ANSYS Fluent to predict the solution variable values for the next time step using a Taylor series expansion, and then inputs that predicted value as an initial guess for the inner iterations of the current time step. As a result, the absolute residual levels are lowered.

   Note that the **Extrapolate Variables** option is not available if you are employing either the NITA scheme with the pressure-based solver or the explicit formulation with the density-based solver.

   Important

   If you use the **Extrapolate Variables** option when modeling an incompressible flow with the density-based solver, it is recommended that you disable the extrapolation of pressure values. After you have enabled the **Extrapolate Variables** option, type the following text command in the console:

```
> solve/set/extrapolate-eqn-vars/pressure

Extrapolate Pressure? [yes] no
```

9. (optional) If you want ANSYS Fluent to gather data for time statistics (that is, time-averaged, root-mean-square, and root-mean-square-error values for solution variables) during the calculation, follow these steps:

   a. Create a custom field function for the each of the variables for which you want to postprocess unsteady statistics (for example, P*|V|), using the **Custom Field Function Calculator**. For detailed instructions, see Creating a Custom Field Function (p. 2004). Note that you do not need to take the extra step of creating custom field functions for the flow shear stresses, flow heat fluxes, wall statistics, or discrete phase variables as there are options for selecting these variables directly in a later step.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parameters & Customization → Custom Field Functions → New...

Important

The maximum number of custom field functions that can be calculated and postprocessed for unsteady statistics is 50.

b. Enable the **Data Sampling for Time Statistics** option in the Run Calculation Task Page (p. 2537).

Solution → Run Calculation → Data Sampling for Time Statistics

Enabling this option will allow you to display and report the mean, the root-mean-square (RMS), and the root-mean-square-error (RMSE) values, as described in Postprocessing for Time-Dependent Problems (p. 1622).

To specify the sampling interval, enter a value for **Sampling Interval**.

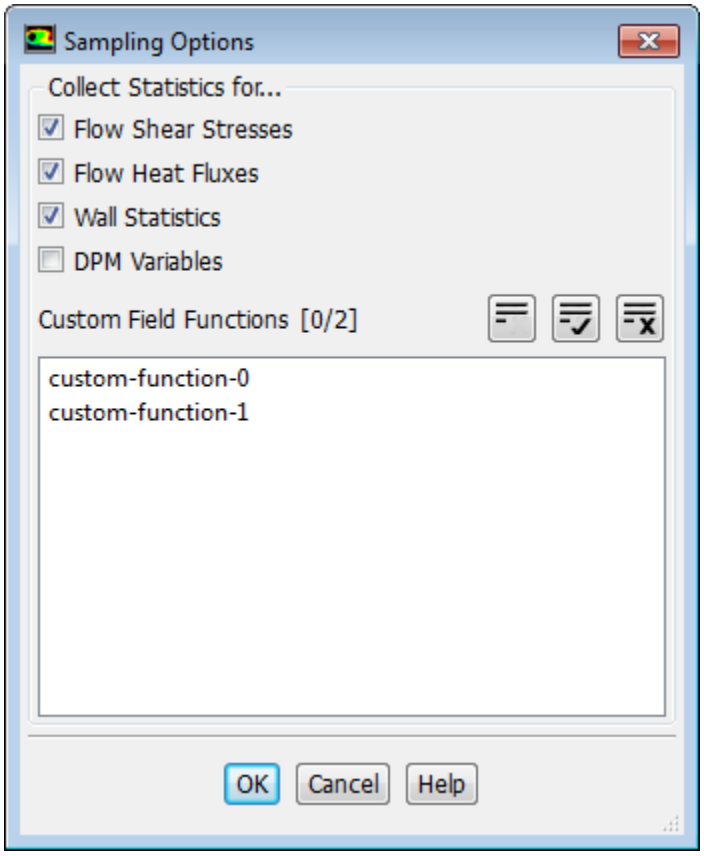
The **Time Sampled** displays the time period over which data has been sampled for the postprocessing of the mean, RMS, and RMSE values. For most quantities, as long as the time step size has been constant, dividing this by the time step size yields the number of data sets that have been collected. However, special consideration is required in the case of discrete phase quantities. As noted in Reporting of Unsteady DPM Statistics (p. 1353), sampling of DPM quantities occurs only when parcels pass through a cell, not necessarily at each specified sampling interval. Therefore, the number of samples of the DPM quantities taken during the sampled time period is stored in the separate postprocessing variable, **Accum DPM Parcels in Cell**. If the time step size is varied, every contribution of data sets sampled is automatically weighted by the current time step size.

To select the variables (which may be represented as custom field functions) for which you want to collect statistics, click the **Sampling Options...** button and make selections from the **Sampling Options** dialog box that opens (Figure 28.27: The Sampling Options Dialog Box (p. 1615)). Note that no custom field functions are selected by default.

Solution → Run Calculation → Sampling Options...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.27: The Sampling Options Dialog Box



Important

Note that gathering data for time statistics is not meaningful inside a moving cell zone (for example, a sliding zone in a sliding mesh problem, a moving zone in a dynamic mesh problem).

c. Initialize the flow statistics.

Solution → **Initialization** **Reset Statistics**

Note that you can also reset the flow statistics after you have gathered some data for time statistics. If you perform, say, 10 time steps with the **Data Sampling for Time Statistics** option enabled, check the results, and then continue the calculation for 10 more time steps, the time statistics will include the data gathered in the first 10 time steps unless you reinitialize the flow statistics.

10. Specify time-dependent solution parameters and start the calculation as described below for the implicit and explicit transient formulations:

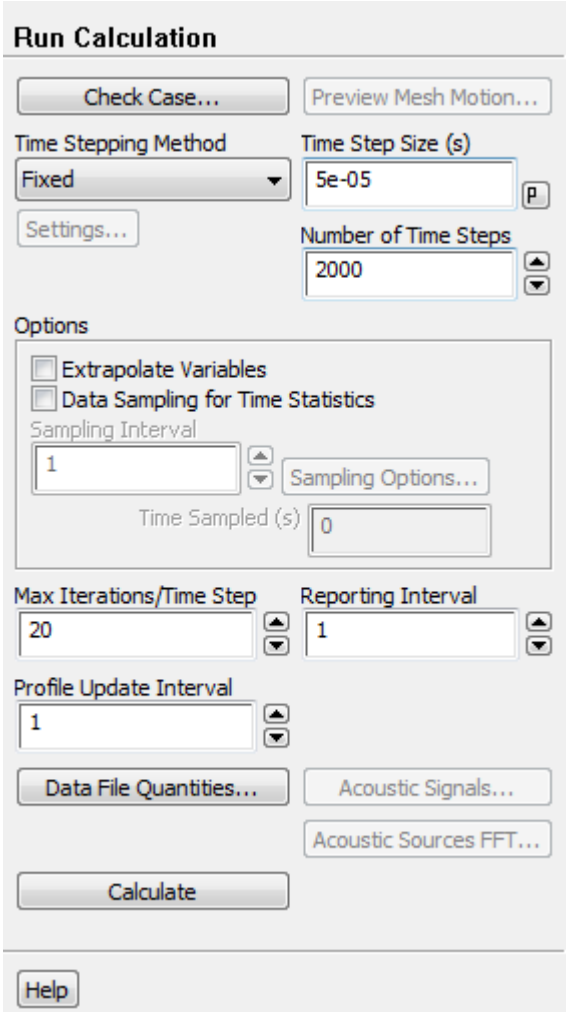
- If you have chosen the **First Order Implicit**, **Second Order Implicit**, or **Bounded Second Order Implicit** formulation, the procedure is as follows:

  a. Set the time-dependent solution parameters in the Run Calculation Task Page (p. 2537) (see Figure 28.28: The Run Calculation Task Page for Implicit Transient Calculations (p. 1616)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution → Run Calculation

Figure 28.28: The Run Calculation Task Page for Implicit Transient Calculations



Solution parameters for the implicit transient formulations are as follows:

– **Max Iterations/Time Step**: When ANSYS Fluent solves the time-dependent equations using the implicit formulation, multiple iterations may be necessary at each time step. This parameter sets a maximum for the number of iterations per time step. If the convergence criteria are met before this number of iterations is performed, the solution will advance to the next time step.

– **Time Step Size**: The time step size is the magnitude of Δt. Since the ANSYS Fluent formulation is fully implicit, there is no stability criterion that must be met in determining Δt. However, to model transient phenomena properly, it is necessary to set Δt at least one order of magnitude smaller than the smallest time constant in the system being modeled. A good way to judge the choice of Δt is to observe the number of iterations ANSYS Fluent needs to converge at each time step. The ideal number of iterations per time step is 5–10. If ANSYS Fluent needs substantially more, the time step is too large. If ANSYS Fluent needs only a few iterations per time step, Δt should be increased. Frequently a time-dependent problem has a very fast "startup" transient that decays rapidly. Therefore, it is often wise to choose a conservatively small Δt for the first 5–10 time steps. Δt may then be gradually increased as the calculation proceeds.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For time-periodic calculations, you should choose the time step based on the time scale of the periodicity. For a rotor/stator model, for example, you might want 20 time steps between each blade passing. For vortex shedding, you might want 20 steps per period.

To verify that your choice for Δt was proper after the calculation is complete, you can plot contours of the Courant number within the domain. To do so, select **Velocity...** and **Cell Convective Courant Number** from the **Contours of** drop-down lists in the **Contours** dialog box. For a stable, efficient calculation, the Courant number should not exceed a value of 20–40 in most sensitive transient regions of the domain.

– **Time Stepping Method**: By default, the size of the time step is fixed (as indicated by the selection of **Fixed**).

To have ANSYS Fluent modify the size of the time step as the calculation proceeds, select **Adaptive** and click the **Settings...** button to specify the parameters in the **Adaptive Time Step Settings** dialog box. See Adaptive Time Stepping (p. 1618) for details.

For transient volume of fluid (VOF) or Eulerian Multiphase calculations you can select the **Variable** time stepping method. The parameters set through the **Parameters...** button are in many ways the same as for the adaptive time stepping method, with the exception of specifying a global Courant number (see Variable Time Stepping (p. 1621)).

Note that with the **Adaptive** or **Variable** time stepping method, the value you specify for the **Time Step Size** will be the initial size of the time step. As the calculation proceeds, the **Time Step Size** shown in the **Run Calculation** task page will be the size of the *current* time step.

b. Specify the desired **Number of Time Steps** in the **Run Calculation** task page and click **Calculate**.

As it calculates a solution, ANSYS Fluent will print the current time at the end of each time step.

- If you choose the **Explicit** transient formulation, you have two input options available for specifying the transient solution advancement. You can specify the transient advancement either directly by specifying a time step size ($\triangle t$) or indirectly via entering a Courant number value (CFL).

a. Under the **Solution Control** panel , if the default option `Specified Time Step` is selected, you will be required to specify the time step size and the `Number of Time Steps` in the **Run Calculation** task page.

Alternatively, if you do not select the `Specified Time Step` option, you will be required to specify the `Courant Number` under the **Solution Control** task page and the `Number of Iterations` in the **Run Calculation** task page. In this option the (CFL) value is used to calculate the time step size ($\triangle t$) using Equation 28.1 (p. 1617):

$$\triangle t = \frac{2 * CFL * V}{\sum_f \lambda_f^{\max} * A_f} \tag{28.1}$$

where, V is the volume of the cell, A_f is the face area, and $\lambda_f^{\max}$ is the maximum local eigenvalue defined in Equation 21.76 in the *Fluent Theory Guide*.

b. Under the **Run Calculation** task page:

– If the Specified Time Step option is selected in the **Solution Control** panel, you need to specify the Time Step Size and the Number of Time Steps.

→ **Time Step Size**: For this option, you have to set the magnitude of $\triangle t$.

> **Note**
>
> The stability of an explicit formulation is limited by the Courant-Fredrichs-Lewy condition. Fluent checks the specified time step against the stability limit. The stability limit is the maximum allowable time step size calculated based on CFL=1 in Equation 28.1 (p. 1617). The stability limit is also the minimum of all local time steps in the domain. If the specified time step size is larger than the maximum allowable time step, Fluent displays a warning during the calculation.

→ **Number of Time Steps**: Specify the desired number of time steps and click **Calculate**

– If the Specified Time Step option is not selected in the **Solution Control** panel then you need to specify the desired Number of Iterations and click **Calculate**.

For explicit transient formulations, every iteration is a time step. The output for results in the Fluent console will include a column indicating the current time (if you have enabled the printing of the current time under adaptive time stepping method).

- You can access the information saved in a data file, which includes a standard set of quantities that were computed during the calculation, by clicking the **Data File Quantities...** button. More information about this feature is available in Setting Data File Quantities (p. 107).

11. Save the final data file (and case file, if you have modified it) so that you can continue the transient calculation later, if desired.

File → **Write** → **Data...**

28.14.1.1. Additional Inputs

The procedures for setting the reporting interval, updating user-defined function profiles, interrupting iterations, and resetting data are the same as those for steady-state calculations. See Performing Steady-State Calculations (p. 1600) for details.

> **Important**
>
> If you are using a user-defined function in your time-dependent calculation, note that, in addition to being updated after every n iterations (where n is the value of the **Profile Update Interval**), the function will also be updated at the first iteration of each time step.

28.14.2. Adaptive Time Stepping

As mentioned in User Inputs for Time-Dependent Problems (p. 1609), it is possible to have the size of the time step change as the calculation proceeds, rather than specifying a fixed size for the entire calculation. This section provides a brief description of the algorithm that ANSYS Fluent uses to compute the time

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

step size, as well as an explanation of each of the parameters that you can set to control the adaptive time stepping.

Important

Adaptive time stepping is available only with the pressure-based and density-based implicit formulations; it cannot be used with the density-based explicit formulation. In addition, it cannot be used with the discrete phase model, second-order time integration, or Euler-Euler multiphase models (Approaches to Multiphase Modeling in the Theory Guide).

28.14.2.1. The Adaptive Time Stepping Algorithm

The automatic determination of the time step size is based on the estimation of the truncation error associated with the time integration scheme. If the truncation error is smaller than a specified tolerance, the size of the time step is increased; if the truncation error is greater, the time step size is decreased.

An estimation of the truncation error can be obtained by using a predictor-corrector type of algorithm [39] (p. 2889) in association with the time integration scheme. At each time step, a predicted solution can be obtained using a computationally inexpensive explicit method (forward Euler for the first-order unsteady formulation, Adams-Bashford for the second-order unsteady formulation). This predicted solution is used as an initial condition for the time step, and the correction is computed using the nonlinear iterations associated with the implicit (pressure-based or density-based) formulation. The norm of the difference between the predicted and corrected solutions is used as a measure of the truncation error. By comparing the truncation error with the desired level of accuracy (that is, the truncation error tolerance), ANSYS Fluent is able to adjust the time step size by increasing it or decreasing it.

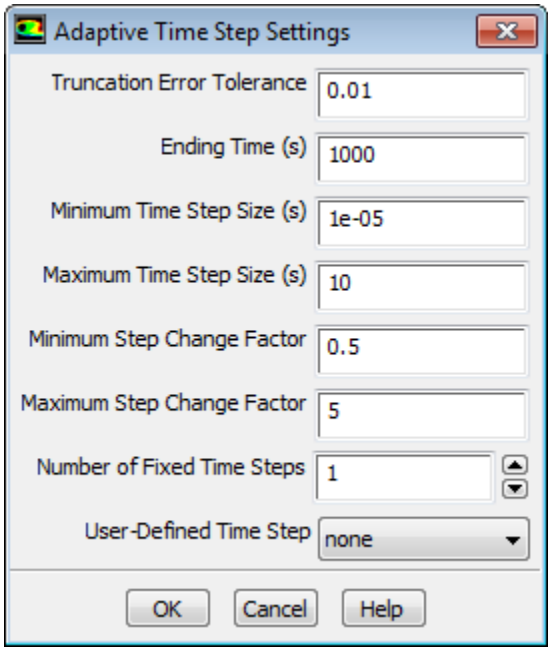
In cases where the truncation error remains above the specified tolerance, ANSYS Fluent will try to meet the tolerance within 5 attempts. If this tolerance is met, then the iteration moves on to the next time step. An explicit scheme is used to predict the solution at each time step, then the explicit prediction is corrected with an implicit scheme. The truncation error, which is a function of the difference between the predicted and corrected solutions at a specific time is used to calculate the next time step. However, if the calculated truncation error is greater than the tolerance limit, we have the option of reverting from the currently performed iteration, which is moving from the n^{th} step to $n+1^{\text{th}}$ step, and performing the iteration with a smaller time step. Note that this option is not available for moving deforming meshes, sliding meshes, and the discrete phase model. Since the truncation error is proportional to the time step, decreasing the time step reduces the truncation error. This can be done until the truncation error goes below the tolerance limit.

28.14.2.2. Specifying Parameters for Adaptive Time Stepping

The parameters that control the adaptive time stepping appear in the **Adaptive Time Step Settings** dialog box, as described in User Inputs for Time-Dependent Problems (p. 1609).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.29: The Adaptive Time Step Settings Dialog Box for Implicit Unsteady Calculations and Adaptive Time Stepping



These parameters are as follows:

Truncation Error Tolerance

specifies the threshold value to which the computed truncation error is compared. Increasing this value will lead to an increase in the size of the time step and a reduction in the accuracy of the solution. Decreasing it will lead to a reduction in the size of the time step and an increase in the solution accuracy, although the calculation will require more computational time. For most cases, the default value of 0.01 is acceptable.

Ending Time

specifies an ending time for the calculation. Since the ending time cannot be determined by multiplying the number of time steps by a fixed time step size, you need to specify it explicitly.

Minimum/Maximum Time Step Size

specify the upper and lower limits for the size of the time step. If the time step becomes very small, the computational expense may be too high; if the time step becomes very large, the solution accuracy may not be acceptable to you. You can set the limits that are appropriate for your simulation.

Minimum/Maximum Step Change Factor

limit the degree to which the time step size can change at each time step. Limiting the change results in a smoother calculation of the time step size, especially when high-frequency noise is present in the solution. If the time step change factor, f, is computed as the ratio between the specified truncation error tolerance and the computed truncation error, the size of time step Δt_n is computed as follows:

- If $1<f<f_{max}$, Δt_n is increased to meet the desired tolerance.
- If $1<f_{max}<f$, Δt_n is increased, but its maximum possible value is $f_{max}\Delta t_{n-1}$.
- If $f_{min}<f<1$, Δt_n is unchanged.
- If $f<f_{min}<1$, Δt_n is decreased.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number of Fixed Time Steps

specifies the number of fixed-size time steps that should be performed before the size of the time step starts to change. The size of the fixed time step is the value specified for **Time Step Size** in the **Run Calculation** task page.

It is a good idea to perform a few fixed-size time steps before switching to the adaptive time stepping. Sometimes spurious discretization errors can be associated with an impulsive start in time. These errors are dissipated during the first few time steps, but they can adversely affect the adaptive time stepping and result in extremely small time steps at the beginning of the calculation.

Important

When the solution tends to exhibit incomplete convergence, rather than increasing the time step size or keeping the same time step size in the next step, ANSYS Fluent reduces the time step size by at least half for the next time step (making sure that the time step size does not go below the specified minimum time step size.

28.14.2.3. Specifying a User-Defined Time Stepping Method

If you want to use your own adaptive time stepping method, instead of the method described above, you can create a user-defined function for your method and select it in the **User-Defined Time Step** drop-down list. The other inputs in the **Adaptive Time Step Settings** dialog box will not be used when you select a user-defined function.

See the Fluent Customization Manual for details about creating and using user-defined functions.

28.14.3. Variable Time Stepping

For VOF and Eulerian multiphase calculations, ANSYS Fluent allows you to use variable time stepping in order to automatically change the time step when an interface is moving through dense cells or if the interface velocity is high.

Variable time stepping is available for both the explicit and the implicit volume fraction formulations when using the VOF model. For the Eulerian multiphase model it is only available when using the Multi-Fluid VOF option or when using the explicit volume fraction formulation.

28.14.3.1. The Variable Time Stepping Algorithm

The global time step Δt_{global} is changed in the following manner:

$$\Delta t_{global} = \frac{CFL_{global}}{max\left(\sum \frac{outgoing\ \ fluxes}{volume}\right)} \tag{28.2}$$

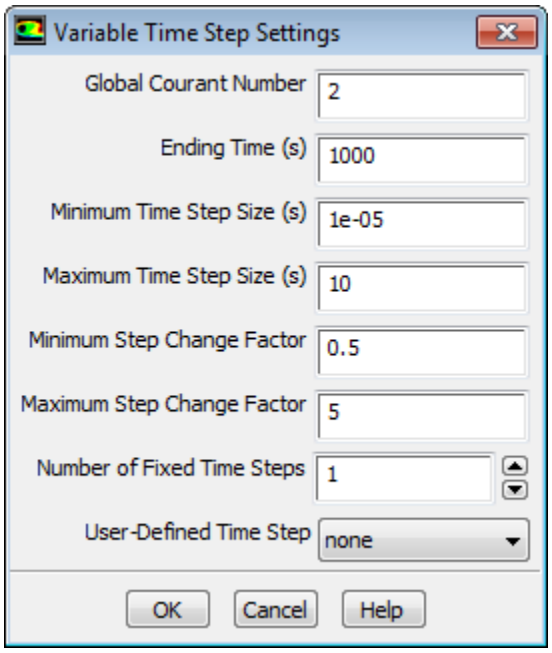
where the ratio $\sum \frac{outgoing\ \ fluxes}{volume}$ is calculated for each cell. ANSYS Fluent takes the maximum of this ratio to calculate the global time step.

28.14.3.2. Specifying Parameters for Variable Time Stepping

For transient VOF calculations, when **Variable** is selected from the **Time Stepping Method** drop-down list, in the **Run Calculation** task page and the **Settings...** button is clicked, the **Variable Time Step Settings** dialog box will open (Figure 28.30: The Variable Time Step Settings Dialog Box for Implicit Unsteady Calculations and Variable Time Stepping (p. 1622)). With the exception of the **Global Courant**

Number field, all parameters are the same as for adaptive time stepping (see Adaptive Time Stepping (p. 1618)). The default value for the **Global Courant Number** is 2.

Figure 28.30: The Variable Time Step Settings Dialog Box for Implicit Unsteady Calculations and Variable Time Stepping



The variable time step is based on the maximum Courant number near the VOF interface. To calculate that Courant number, ANSYS Fluent uses a flux-based definition where, in the region near the fluid interface, ANSYS Fluent divides the volume of each cell by the sum of the outgoing fluxes. The resulting time represents the time it would take for the fluid to empty out of the cell. The smallest such time is used as the characteristic time of transit for a fluid element across a control volume.

Note

When choosing the values for Minimum and Maximum Step Change Factors, it is generally not recommended to allow large variation in time step size due to stability considerations. This is especially true when time step size is ramping up.

Variable time stepping is not ideally suited to second order time formulation because the second order formulation does not take into account the time step size variation. However, by keeping the minimum time step change factor in the range 0.5–0.8 and the maximum time step change factor in the range 1.2–1.5 it can be used without severely affecting the second order time accuracy.

28.14.4. Postprocessing for Time-Dependent Problems

The postprocessing of time-dependent data is similar to that for steady-state data, with all graphical and alphanumeric commands available. You can read a data file that was saved at any point in the calculation (by you or with the autosave option) to restore the data at any of the time levels that were saved.

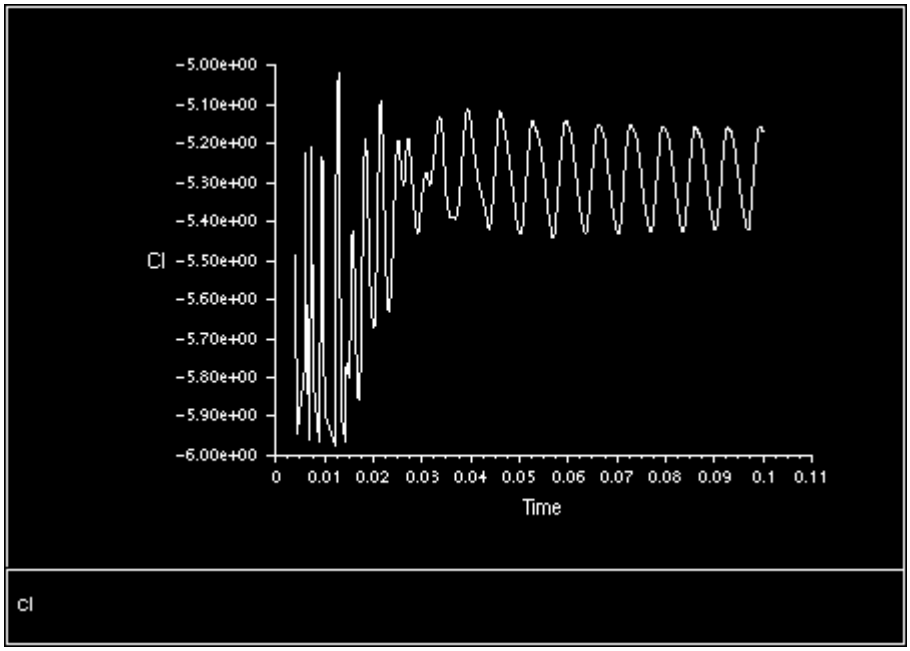
File → Read → Data...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS Fluent will label any subsequent graphical or alphanumeric output with the time value of the current data set.

If you save data from the force or surface report definitions to files (see step 5 in User Inputs for Time-Dependent Problems (p. 1609)), you can read these files back in and plot them to see a time history of the monitored quantity. Figure 28.31: Lift Coefficient Plot for a Time-Periodic Solution (p. 1623) shows a sample plot generated in this way.

Figure 28.31: Lift Coefficient Plot for a Time-Periodic Solution



If you enable the **Data Sampling for Time Statistics** option in the Run Calculation Task Page (p. 2537) (see Performing Time-Dependent Calculations (p. 1608) for details), ANSYS Fluent will compute the time average (mean) of the instantaneous values, the root-mean-square, and the root-mean-square-errors of those quantities and custom field functions that are enabled/selected in the **Sampling Options** dialog box.

The **Time Sampled** displays the time over which data has been sampled for the postprocessing of the mean, RMS, and RMSE values. For most quantities, as long as the time step size has been constant, dividing this by the time step size yields the number of data sets that have been collected. However, special consideration is required In the case of discrete phase quantities. As noted in Reporting of Unsteady DPM Statistics (p. 1353), sampling of DPM quantities occurs only when parcels pass through a cell, not necessarily at each specified sampling interval. Therefore, the number of samples of the DPM quantities taken during the sampled time period is stored in the separate postprocessing variable, **Accum DPM Parcels in Cell**. If the time step size is varied, every contribution of data sets sampled is automatically weighted by the current time step size.

The mean, root-mean-square(RMS), and root-mean-square-error (RMSE) values for solution variables will be available in the **Unsteady Statistics...** category (or the **Unsteady DPM Statistics...** category for discrete phase quantities) of the variable selection drop-down list that appears in postprocessing dialog boxes. For example, in the **Contours** dialog box, you could select **Unsteady Statistics...** and **RMSE-uns-custom-functon-0** for the **Contours of** drop-down lists in order to display the root-mean-square-errors of a custom field function named **uns-custom-functon-0**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.15. Monitoring Solution Convergence

During the solution process you can monitor the convergence dynamically by checking residuals, statistics, force values, surface integrals, and volume integrals. You can print reports of or display plots of lift, drag, and moment coefficients, surface integrations, and residuals for the solution variables. For unsteady flows, you can also monitor elapsed time. Creating report definitions for monitoring is described in Creating Report Definitions (p. 1886).

For additional information, see the following sections:

28.15.1. Monitoring Residuals
28.15.2. Monitoring Statistics
28.15.3. Monitoring Solution Quantities

28.15.1. Monitoring Residuals

At the end of each solver iteration, the residual sum for each of the conserved variables is computed and stored, thereby recording the convergence history. This history is also saved in the data file. The residual sum is defined below.

On a computer with infinite precision, these residuals will go to zero as the solution converges. On an actual computer, the residuals decay to some small value ("round-off") and then stop changing ("level out"). For single-precision computations (the default for workstations and most computers), residuals can drop as many as six orders of magnitude before hitting round-off. Double-precision residuals can drop up to twelve orders of magnitude. Guidelines for judging convergence can be found in Judging Convergence (p. 1668).

28.15.1.1. Definition of Residuals for the Pressure-Based Solver

After discretization, the conservation equation for a general variable ϕ at a cell P can be written as

$$a_P \phi_P = \sum_{nb} a_{nb} \phi_{nb} + b \tag{28.3}$$

Here a_P is the center coefficient, a_{nb} are the influence coefficients for the neighboring cells, and b is the contribution of the constant part of the source term S_c in $S = S_c + S_P \phi$ and of the boundary conditions. In Equation 28.3 (p. 1624),

$$a_P = \sum_{nb} a_{nb} - S_P \tag{28.4}$$

The residual R^{ϕ} computed by ANSYS Fluent's pressure-based solver is the imbalance in Equation 28.3 (p. 1624) summed over all the computational cells P. This is referred to as the "unscaled" residual. It may be written as

$$R^{\phi} = \sum_{cells\ P} \left| \sum_{nb} a_{nb} \phi_{nb} + b - a_P \phi_P \right| \tag{28.5}$$

In general, it is difficult to judge convergence by examining the residuals defined by Equation 28.5 (p. 1624) since no scaling is employed. This is especially true in enclosed flows such as natural convection in a room where there is no inlet flow rate of ϕ with which to compare the residual.

ANSYS Fluent scales the residual using two kinds of scaling factors, representative of the flow rate of ϕ through the domain. The factors are termed *global scaling* and *local scaling*. The type of scaling can be selected from the **Residual Monitors** dialog box. The "globally scaled" residual is defined as

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$R^{\phi}=\frac{\sum_{\text{cells } P}\left|\sum_{nb}a_{nb}\phi_{nb}+b-a_P\phi_P\right|}{\sum_{\text{cells } P}\left|a_P\phi_P\right|} \tag{28.6}$$

For the momentum equations the denominator term $a_P\phi_P$ is replaced by $a_P v_P$, where v_P is the magnitude of the velocity at cell P.

The "locally scaled" residual is defined as

$$R_{\phi}=\frac{\sqrt{\sum_{\text{cells}}^{n}\left(\frac{1}{n}\right)\left(\frac{\sum_{nb}a_{nb}\phi_{nb}+b-a_p\phi_p}{a_p}\right)^2}}{\left(\phi_{max}-\phi_{min}\right)_{domain}} \tag{28.7}$$

As in global scaling, for the momentum, ϕ is replaced by v of the cell.

The scaled residual is a more appropriate indicator of convergence for most problems. The selection of scaling and convergence criteria for different types of scaling are discussed in Judging Convergence (p. 1668). The default residuals displayed by ANSYS Fluent are global scaling.

For the continuity equation, the unscaled residual for the pressure-based solver is defined as

$$R^{c}=\sum_{\text{cells } P}\left|\text{rate of mass creation in cell P}\right| \tag{28.8}$$

The local scaling is the same for all equations. However, the global scaling treats continuity in a different way and it is defined as

$$\frac{R^{c}_{\text{iteration } N}}{R^{c}_{\text{iteration } 5}} \tag{28.9}$$

The denominator is the largest absolute value of the continuity residual in the first five iterations.

The scaled residuals described above are useful indicators of solution convergence. Guidelines for their use are given in Judging Convergence (p. 1668). It is sometimes useful to determine how much a residual has decreased during calculations as an additional measure of convergence. For this purpose, ANSYS Fluent allows you to normalize the residual (either scaled or unscaled) by dividing by the maximum residual value after M iterations, where M is set by you in the Residual Monitors Dialog Box (p. 2789) in the **Iterations** field under **Residual Values**.

$$\overline{R}^{\phi}=\frac{R^{\phi}_{\text{iteration } N}}{R^{\phi}_{\text{iteration } M}} \tag{28.10}$$

Normalization in this manner ensures that the initial residuals for all equations are of $O(1)$ and is sometimes useful in judging overall convergence.

By default, $M=5$. You can also specify the normalization factor (the denominator in Equation 28.10 (p. 1625)) manually in the Residual Monitors Dialog Box (p. 2789).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.15.1.2. Definition of Residuals for the Density-Based Solver

A residual for the density-based solver is simply the time rate of change of the conserved variable (W). The RMS residual is the square root of the average of the squares of the residuals in each cell of the domain:

$$R(W)=\sqrt{\sum\left(\frac{\partial W}{\partial t}\right)^2} \tag{28.11}$$

Equation 28.11 (p. 1626) is the unscaled residual sum reported for all the coupled equations solved by ANSYS Fluent's density-based solver.

Important

The residuals for the equations that are solved sequentially by the density-based solver (turbulence and other scalars, as discussed in Density-Based Solver in the Theory Guide) are the same as those described above for the pressure-based solver.

In general, it is difficult to judge convergence by examining the residuals defined by Equation 28.11 (p. 1626) since no scaling is employed. This is especially true in enclosed flows such as natural convection in a room where there is no inlet flow rate of ϕ with which to compare the residual. As with the pressure-based solver, ANSYS Fluent uses two types of scaling for the density-based solver. The globally scaled residual is defined as

$$\frac{R(\mathrm{W})_{\text{iteration } N}}{R(\mathrm{W})_{\text{iteration } 5}} \tag{28.12}$$

The denominator is the largest absolute value of the residual in the first five iterations.

The locally scaled residual is calculated from the local flux imbalance in the cell. It is calculated using Equation 28.13 (p. 1626):

$$R_\phi=\frac{\sqrt{\sum_{n}^{cells}\left(\frac{1}{n}\right)\left(\frac{\partial w}{\partial t}V\right)^2}}{(w_{max}-w_{min})_{domain}} \tag{28.13}$$

w is the conservative variable and V is the cell volume. The unscaled residual is always calculated from Equation 28.11 (p. 1626).

The scaled residuals described above are useful indicators of solution convergence. Guidelines for their use are given in Judging Convergence (p. 1668). It is sometimes useful to determine how much a residual has decreased during calculations as an additional measure of convergence. For this purpose, ANSYS Fluent allows you to normalize the residual (either scaled or unscaled) by dividing by the maximum residual value after M iterations, where M is set by you in the Residual Monitors Dialog Box (p. 2789) in the **Iterations** field under **Residual Values**.

Normalization of the residual sum is accomplished by dividing by the maximum residual value after M iterations, where M is set by you in the Residual Monitors Dialog Box (p. 2789) in the **Iterations** field under **Residual Values**:

$$\overline{R}(\mathrm{W})=\frac{R(\mathrm{W})_{\text{iteration } N}}{R(\mathrm{W})_{\text{iteration } M}} \tag{28.14}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Normalization in this manner ensures that the initial residuals for all equations are of $O(1)$ and is sometimes useful in judging overall convergence.

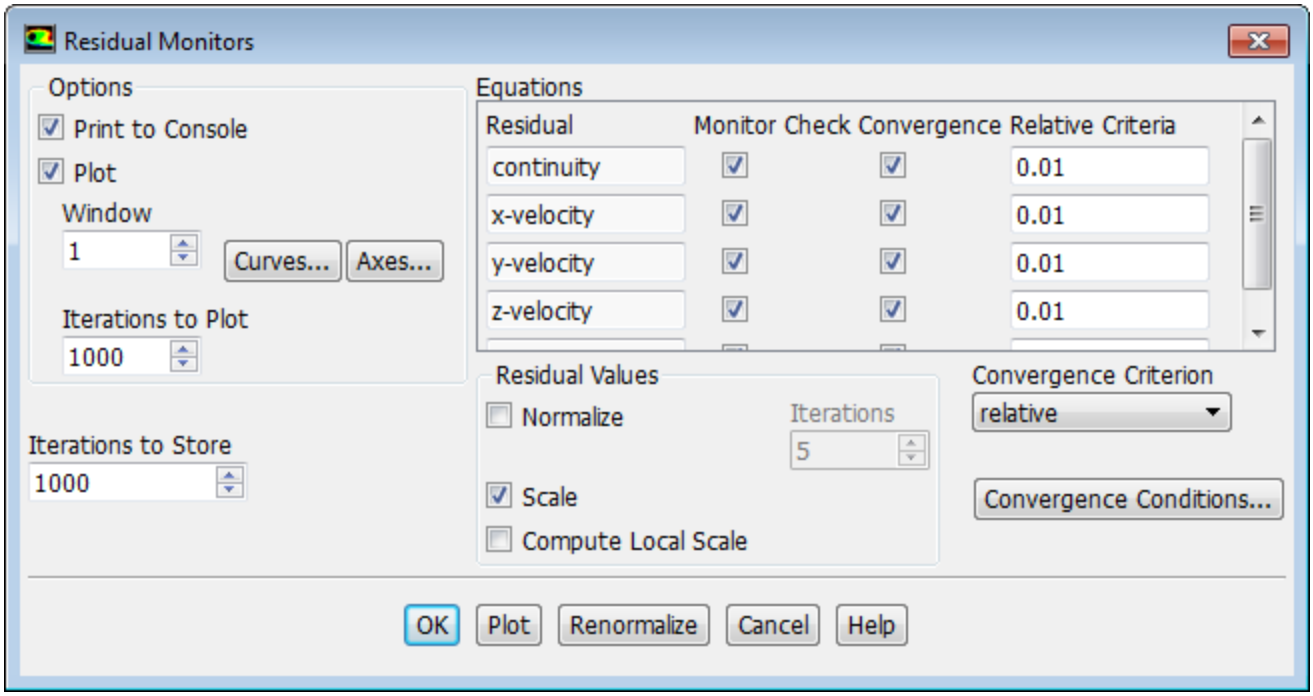
By default, $M=5$, making the normalized residual equivalent to the scaled residual. You can also specify the normalization factor (the denominator in Equation 28.14 (p. 1626)) manually in the Residual Monitors Dialog Box (p. 2789).

28.15.1.3. Overview of Using the Residual Monitors Dialog Box

All inputs controlling the monitoring of residuals are entered using the Residual Monitors Dialog Box (p. 2789)(Figure 28.32: The Residual Monitors Dialog Box (p. 1627)).

Solution → Monitors → Residuals Edit...

Figure 28.32: The Residual Monitors Dialog Box



In general, you will only need to enable residual plotting and modify the convergence criteria using this dialog box. Additional controls are available for disabling monitoring of particular residuals, and modifying normalization and plot parameters.

28.15.1.4. Printing and Plotting Residuals

By default, residual values for all relevant variables are printed in the console after each iteration. If you want to disable this printout, turn off **Print to Console** under **Options**. To enable the plotting of residuals after each iteration, turn on **Plot** under **Options**. Residuals will be plotted in the graphics window (with the window ID set in the **Window** field) during the calculation.

If you want to display a plot of the current residual history, simply click the **Plot** push button.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.15.1.5. Storing Residual History Points

Residual histories for each variable are automatically saved in the data file, regardless of whether they are being monitored. You can control the number of history points to be stored by changing the **Iterations to Store** entry. By default, up to 1000 points will be stored. If more than 1000 iterations are performed (that is, the limit is reached), every other point will be discarded—leaving 500 history points—and the next 500 points will be stored. When the total hits 1000 again, every other point will again be discarded, and so on. If you are performing a large number of iterations, you will lose a great deal of residual history information at the beginning of the calculation. In such cases, you should increase the **Iterations to Store** value to a more appropriate value. Of course, the larger this number is, the more memory you will need, the longer the plotting will take, and the more disk space you will need to store the data file.

28.15.1.6. Controlling Normalization and Scaling

By default, scaling of residuals (see Equation 28.6 (p. 1625) and Equation 28.12 (p. 1626)) is enabled and the default convergence criterion is 10^{-6} for energy and P-1 equations and 10^{-3} for all other equations. When the **Scale** option is enabled, global scaling will be applied by default. You can then enable the **Compute Local Scale** option and select the **Reporting Option** from the drop-down list. You have a choice to plot or print to the console the **local scaling** or the **global scaling** of residuals. By default, the **global scaling** of residuals will be plotted.

Note

When **Compute Local Scale** is enabled, ANSYS Fluent computes and stores both the locally and globally scaled residuals from subsequent iterations, for the purpose of reporting. The scaled residuals are stored in the data file.

Important

Once the **Compute Local Scale** option is enabled and you disable the **Scale** option, the **Compute Local Scale** option will not automatically be disabled. Instead, it will compute both the locally and globally scaled residuals, but only print or plot the unscaled residual.

Residual normalization (that is, dividing the residuals by the largest value during the first few iterations) is also available but disabled by default.

Normalization can be used with both scaled and unscaled residuals. Note that if normalization is enabled, the convergence criterion may need to be adjusted appropriately. See Judging Convergence (p. 1668) for information about judging convergence based on the different types of residual reports. (Both the raw residuals and scaling factors are stored in the data file, so you can switch between scaled and unscaled residuals.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To report unscaled residuals, simply disable the **Scale** option under **Residual Values**.

Important

If you switch from scaled to unscaled residuals (or vice versa) and you are normalizing the residuals (as described below), you must click the **Renormalize** button to recompute the normalization factors.

If you want to normalize the residuals (see Equation 28.10 (p. 1625) or Equation 28.14 (p. 1626)), enable the **Normalize** option under **Residual Values**. The **Normalization Factor** column will be added to the dialog box at this time. ANSYS Fluent will normalize the printed or plotted residual for each variable by the value indicated as the **Normalization Factor** for that variable. The default **Normalization Factor** is the maximum residual value after the first 5 iterations. To use the maximum residual value after a different number of iterations (that is, specify a different value for M in Equation 28.10 (p. 1625) or Equation 28.14 (p. 1626)), you can modify the **Iterations** entry under **Residual Values**.

In some cases, the maximum residual may occur sometime after the iteration specified in the **Iterations** field. If this should occur, you can click the **Renormalize** button to set the normalization factors for all variables to the maximum values in the residual histories. Subsequent plots and printed reports will use the new normalization factor.

You can also specify the normalization factor (the denominator in Equation 28.10 (p. 1625) or Equation 28.14 (p. 1626)) explicitly. To modify the normalization factor for a particular variable, enter a new value in the corresponding **Normalization Factor** field in the Residual Monitors Dialog Box (p. 2789).

If you want to report unnormalized, unscaled residuals (Equation 28.5 (p. 1624) or Equation 28.11 (p. 1626)), disable the **Normalize** and **Scale** options under **Residual Values** in the Residual Monitors Dialog Box (p. 2789). Note that unnormalized, unscaled residuals are stored in the data file regardless of whether the reported residuals are normalized or scaled.

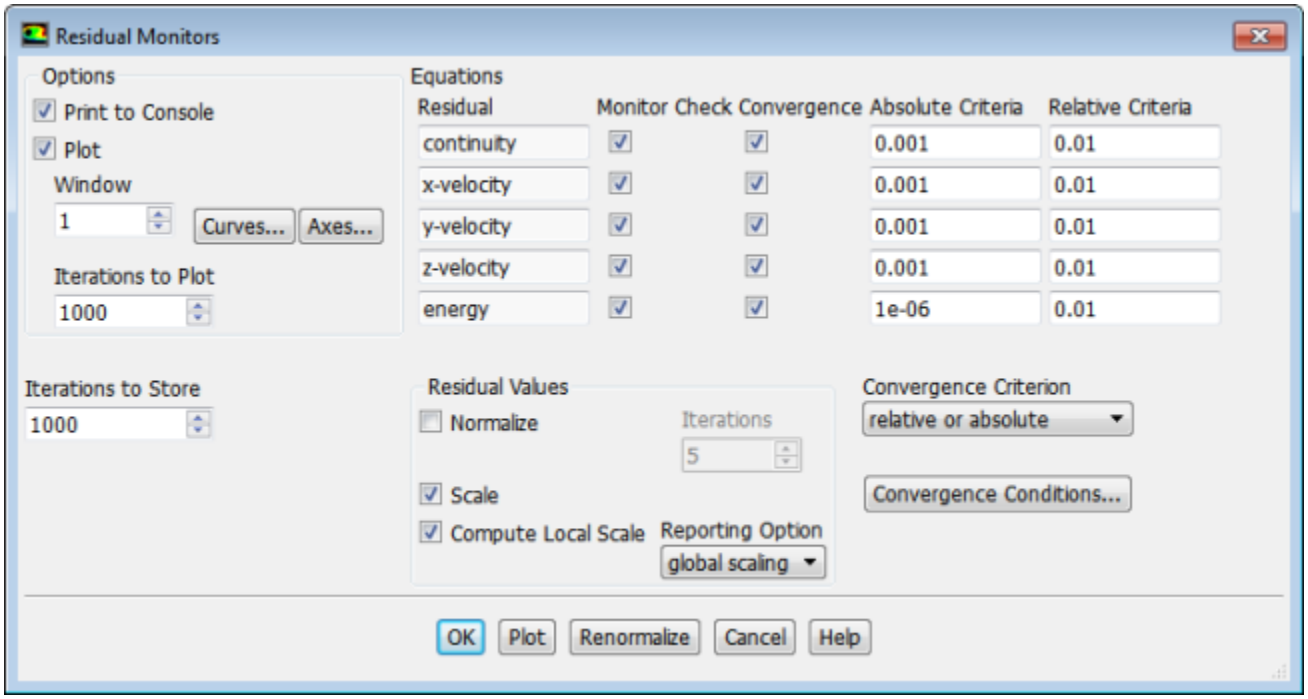
28.15.1.7. Choosing a Convergence Criterion

The ability to choose certain convergence criteria provides you with alternative ways to check convergence when using the iterative transient solver. The various convergence criteria can be selected in the **Residual Monitors** dialog box from the **Convergence Criterion** drop-down list.

Solution → **Monitors** → **Residuals** **Edit...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.33: The Residual Monitors Dialog Box Displaying Relative or Absolute Convergence



Four options are available for checking an equation for convergence:

absolute
This is the default. For steady-state cases, **absolute** and **none** are the only options available for selection. The residual (scaled and/or normalized) of an equation at an iteration is compared with a user-specified value. If the residual is less than the user-specified value, that equation is deemed to have converged for a time step.

relative
The residual of an equation at an iteration of a time step is compared with the residual at the start of the time step. If the ratio of the two residuals is less than a user-specified value, that equation is deemed to have converged for a time step.

relative or absolute
If either the **absolute** convergence criterion or the **relative** convergence criterion is met, the equation is considered converged.

The **Relative Criteria** can be set when **relative** or **relative or absolute** is selected.

none
Convergence checking is disabled.

In many situations, the **absolute** convergence criterion could be too stringent for transient flows causing a large number of iterations per time step. For example, the scaling of the continuity equation is based on the value of the continuity residual in the first five iterations. The scaling factor could be low if the initial continuity residual is small and therefore the scaled residual could fail to meet the **absolute** convergence criterion. With the **relative** convergence criterion, convergence is checked by comparing the residual at an iteration of a time step with the residual at the beginning of the time step and hence this problem is alleviated. The **relative or absolute** convergence criterion is useful in situations where the residuals of some of the equations are already very low at the start of a time step (for example, when a particular variable has reached steady-state), and the order of magnitude reduction in residuals

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

is not possible. The **none** option allows you to disable convergence checking by selecting the option in the **Convergence Criterion** drop-down list.

Important

relative and **relative or absolute** convergence criteria are available only with the unsteady pressure-based solver and unsteady density-based solver.

The text command used to access the convergence criterion is

`solve` → `monitors` → `residual` → `criterion-type`

When `criterion-type` is entered, you will have the following choices:

Criterion	Type
0	absolute
1	relative
2	relative or absolute
3	none

For `criterion-type` 1 or 2, the text command `relative-conv-criteria` will appear under the `residual` text menu, where the various `relative-conv-criteria` can be set.

Important

If the NITA solver is enabled, no convergence criteria are available for selection.

28.15.1.8. Modifying Convergence Criteria

Depending on the **Convergence Criterion** you choose, ANSYS Fluent will check for convergence. If convergence is being monitored, the solution will stop automatically when each variable meets its specified convergence criterion. Convergence checks can only be performed on variables whose residuals are being monitored (variables with the **Monitor** option enabled).

You can choose whether or not you want to check the convergence for each variable by enabling or disabling the **Check Convergence** option for it in the Residual Monitors Dialog Box (p. 2789). To modify the convergence criterion for a particular variable, enter a new value in the corresponding convergence criterion field.

28.15.1.9. Disabling Monitoring

If your problem requires the solution of many equations (for example, turbulence quantities and multiple species), a plot that includes all residuals may be difficult to read. In such cases, you may choose to monitor only a subset of the residuals, perhaps those that affect convergence the most. You can indicate whether or not you want to monitor residuals for each variable by enabling or disabling the relevant check box in the **Monitor** list of the Residual Monitors Dialog Box (p. 2789).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.15.1.10. Plot Parameters

If you choose to plot the residual values (either interactively during the solution or using the **Plot** button after calculations are complete), there are several display parameters you can modify.

In the **Window** field under **Options**, you can specify the ID of the graphics window in which the plot will be drawn. When ANSYS Fluent is iterating, the active graphics window is temporarily set to this window to update the residual plot, and then returned to its previous value. Thus, the residual plot can be maintained in a separate window that does not interfere with other graphical postprocessing.

You can modify the number of residual history points to be displayed in the plot by changing the **Iterations to Plot** entry under **Options**. If you specify n points, ANSYS Fluent will display the last n history points. Since the y axis is scaled by the minimum and maximum values of all points in the plot, you can zoom in on the end of the residual history by setting **Iterations to Plot** to a value smaller than the number of iterations performed. If, for example, the residuals jumped early in the calculation when you turned on turbulence, that peak broadens the overall range in residual values, making the smaller fluctuations later on almost indistinguishable. By setting the value of **Iterations to Plot** so that the plot does not include that early peak, your y-axis range is better suited to the values that you are interested in seeing. For more information on residual history points, refer to the discussion of storing residual history points, described earlier in this section.

You can also modify the attributes of the plot axes and the residual curves. Click the **Axes...** or **Curves...** button to open the Axes Dialog Box (p. 2608) or Curves Dialog Box (p. 2610). See Axes Dialog Box (p. 2608) and Curves Dialog Box (p. 2610) for details.

Important

Note that entering a value for **Iterations to Plot** does not necessarily mean *solved* iterations but rather *stored* (or sampled) data points. Note also that the frequency of the data storage will diminish towards the start of the solution as the number of solved iterations increases. Due to this, whenever the *stored* iterations is greater than the *solved* iterations, if you plot *n* iterations, you actually see a history that goes back further than *n* solved iterations.

28.15.1.11. Postprocessing Residual Values

If you are having solution convergence difficulties, it is often useful to plot the residual value fields (for example, using contour plots) to determine where the high residual values are located. When you use the density-based solver, the residual values for all solution variables are available in the **Residuals...** category in the postprocessing dialog boxes. (If you read case and data files into ANSYS Fluent, you will need to perform at least one iteration before the residual values are available for postprocessing.) For the pressure-based solver, however, only the mass imbalance in each cell is available by default.

Important

The residual values that are made available for postprocessing are the unscaled values.

If you want to plot residual value fields for a pressure-based solver calculation, you will need to do the following:

1. Read in the case and data files of interest (if they are not already in the current session).

2. Use the `expert` command in the `solve/set/` text menu to enable the saving of residual values.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`solve` → `set` → `expert`

Among other questions, ANSYS Fluent will ask if you want to save cell residuals for postprocessing. Enter `yes` or `y`, and keep the default settings for all of the other questions (by pressing the <RETURN> key).

3. Perform at least one iteration.

The solution variables for which residual values are available will appear in the **Residuals...** category in the postprocessing dialog boxes. Note that residual values are *not* available for the radiative transport equations solved by the discrete ordinates radiation model.

28.15.2. Monitoring Statistics

If you are solving a fully-developed periodic flow, you may want to monitor the pressure gradient or the bulk temperature ratio, as discussed in Periodic Flows (p. 582).

If you are solving an unsteady flow (especially if you are using the explicit time stepping option), you may want to monitor the "time" that has elapsed during the calculation. The physical time of the flow field starts at zero when you initialize the flow. (See Performing Time-Dependent Calculations (p. 1608) for details about modeling unsteady flows.)

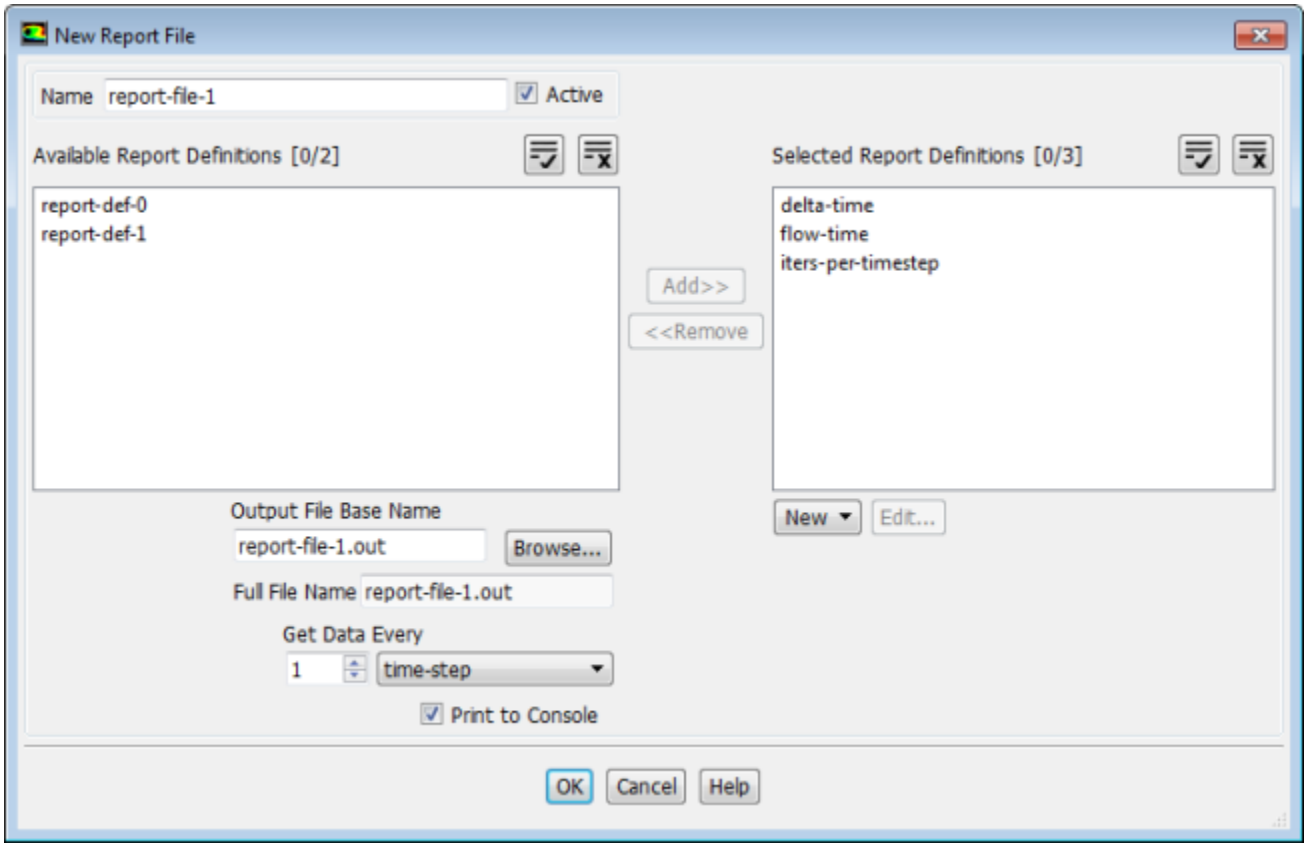
If you are using the adaptive time stepping method described in Adaptive Time Stepping (p. 1618), you may want to monitor the size of the time step, Δt.

You can create report files and plots for each of these quantities, flow-time, delta-time, and iterations-per-timestep, per-pr-grad (periodic pressure gradient), and per/bulk-temp-ratio (periodic bulk temperature ratio) or they can all be included in a single report file; flow-time and delta-time can be included in the same report plot.

Solution → **Monitors** → **Report File** **New...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.34: Report File for 'flow-time', 'delta-time', and 'iters-per-timestep'



The procedure for setting up this monitor is listed below:

1. Indicate the type of report you want by enabling the **Print to Console** option for a printout and/or the **Plot** option for a plot.

2. Select the appropriate quantity in the **Statistics** list.

3. If you want to write the statistics to files, enable **Write**. Each selected statistic will be written to a file named with the base name listed in **File Basename** and the name of the statistic inserted immediately before the period. For example, if **File Basename** is set to `statistics-.out`, the **delta_time** statistic will be written to `statistics-delta_time.out`, the time statistic will be written to `statistics-time.out`, and so on.

4. Select whether to use **Iteration**, **Time Step**, or **Flow Time** for the **X Axis** quantity.

   Note

   The statistic monitors are evaluated, printed, plotted, and written according to the selection for **X Axis** — every iteration if **Iteration** is selected, and every time step otherwise. Thus, if you select **Time Step** or **Flow Time** and subsequently change to a steady-state solution, the monitor will not be evaluated unless you also change the **X Axis** setting back to **Iteration**.

5. If you are plotting the quantities, you can set any of the plotting options discussed below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.15.2.1. Plot Parameters

If you choose to plot the statistics, there are several display parameters you can modify.

In the **Window** field, you can specify the ID of the graphics window in which the plot will be drawn (or in which the first plot will be drawn, if you are plotting more than one quantity.) When ANSYS Fluent is iterating, the active graphics window is temporarily set to this window to update the plot, and then returned to its previous value. Thus, the statistics plot can be maintained in a separate window that does not interfere with other graphical postprocessing. Note that additional quantities that you have selected in the **Statistics** list will be plotted in windows with incrementally higher IDs.

You can also modify the attributes of the plot axes and curves. Click the **Axes...** or **Curves...** button to open the Axes Dialog Box (p. 2608) or Curves Dialog Box (p. 2610). See Axes Dialog Box (p. 2608) and Curves Dialog Box (p. 2610) for details.

28.15.3. Monitoring Solution Quantities

Solution quantities for which Report Definitions have been created can be monitored throughout the solution to aid in judging convergence. You can setup your case file so that force and moment coefficients and surface and volume integrals are computed and stored at the end of every iteration (for steady-state solutions) or time step (for transient solutions), and thereby create a convergence history. You can also choose to compute the value averaged over several iterations/time steps. Running averages can also help you determine if you have reached convergence for a solution that is oscillating or irregular. You can print and plot and save this convergence data to an external file using Report Files and Report Plots. The external file is written in the report file format described in Report Files and Report Plots (p. 1903).

Report Files and Report Plots are accessible under the **Monitors** node in the tree.

Solution → **Monitors** → **Report Files or Report Plots**

For additional information on setting up report definitions and report files and plots, see Monitoring and Reporting Solution Data (p. 1885) and Creating Report Definitions (p. 1886).

28.16. Convergence Conditions

The convergence conditions facility allows you to set convergence conditions on the solution that are based on the values from report definitions (surface, volume, lift, drag, and so on). There are two options for declaring convergence using report definitions:

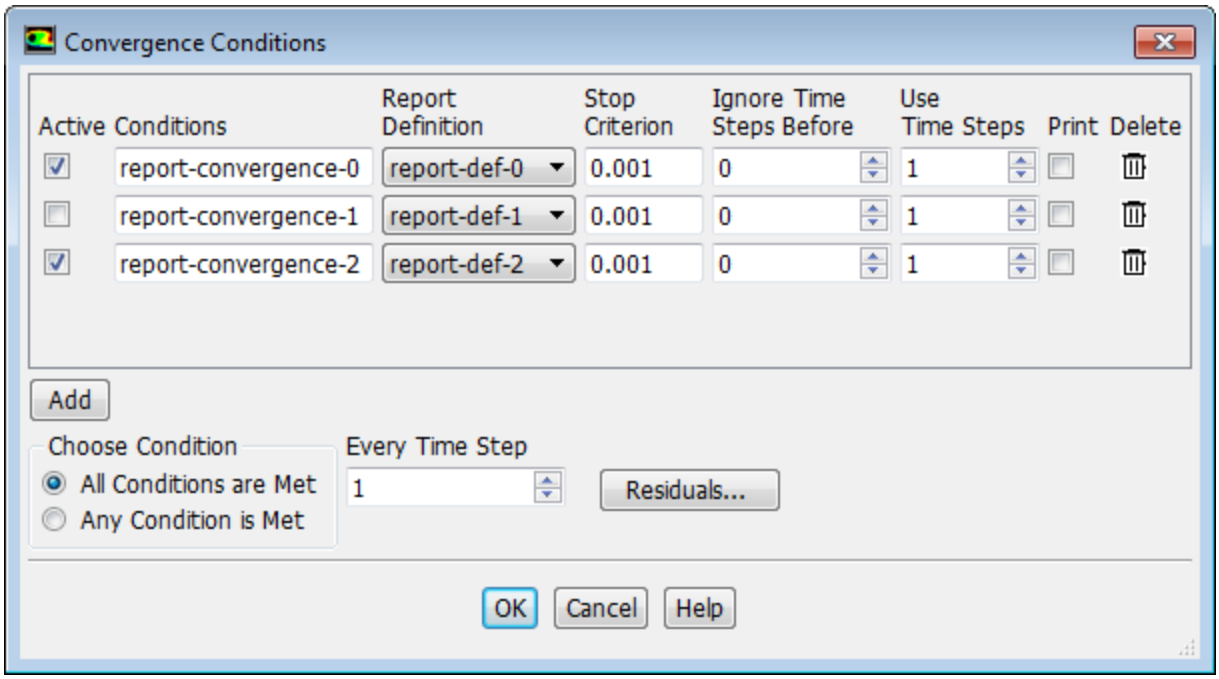
1. The solution is considered to be converged if all of the active report definitions' criteria are satisfied.

2. The solution is considered to be converged if any of the active report definitions' criteria is satisfied.

Convergence check for the report definitions is independent of that for the equation residuals. By default, the solution convergence will happen if either report definition convergence criteria or residuals convergence criteria is satisfied. If you want to rely only on the report definition values to decide convergence, clear the **Check Convergence** check boxes in the **Residual Monitors** dialog box.

To open the **Convergence Conditions** dialog box, click the **Convergence...** button in the **Solving** ribbon tab (**Reports** group box).

Solving → **Reports** → **Convergence...**



The **Active** column of check boxes (on the left) allows you to deactivate/activate convergence conditions.

28.16.1. Setting Up the Convergence Conditions Dialog Box

Create new convergence conditions by clicking **Add** and selecting a **Report Definition** from the drop-down list.

To create meaningful conditions for determining convergence from report definitions, set the levels of the convergence condition variables according to the following directions.

Enter a value in the **Ignore Iterations Before** column if you expect your solution to fluctuate in the first few iterations. Enter a value that represents the number of iterations you anticipate the fluctuations to continue. The calculation will begin after this number of iterations has finished.

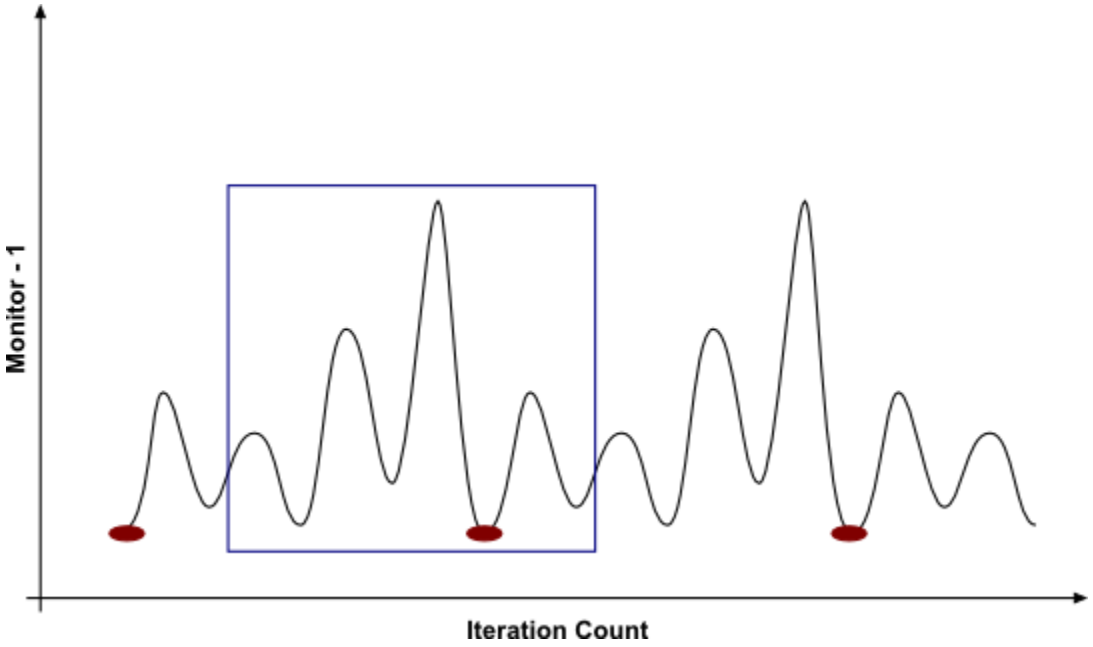
Use the **Use Iterations** setting to select the number of previous iterations to be included in the monitor convergence check. For fluctuating simulations like Figure 28.35: Fluctuating Simulation Example (p. 1637), this number should be high enough to counteract the effect of the fluctuations.

Note

Transient cases use time steps rather than iterations. When solving a transient case, the **Convergence Conditions** dialog box will relabel some fields: **Every Iteration** will change to **Every Time-step**, **Ignore Iterations Before** will change to **Ignore Time Steps Before**, and **Use Iterations** will change to **Use Time Steps**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.35: Fluctuating Simulation Example



The **Stop Criterion** indicates the criterion below which the solution is considered to be converged. The value of **Stop Criterion** is calculated as follows:

Res-m(1) = [abs (m(n) - m(n-1))]/m(n)

Res-m(2) = [abs (m(n) - m(n-2))]/m(n)

Res-m(3) = [abs (m(n) - m(n-3))]/m(n)

.

.

.

Res-m(Np) = [abs (m(n) - m(n-Np))]/m(n)

Where:

Res-m is the report definition residual

n is the iteration number

m(n) is the value of the report definition at the n^{th} iteration

Np is the number of previous iterations to consider (**Use Iterations**/**Use Time Steps**)

The report definition residual is the absolute variance of the monitored quantity over the last Np iterations, divided by the current value of the monitored quantity.

If the maximum value of all of the Res-m values is less than the **Stop Criterion** number, the solution is considered to be converged at the n^{th} iteration. If the maximum value of all of the Res-m values is greater than or equal to the **Stop Criterion** number, the calculation will proceed to the next iteration.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select the **Print** check box to print report definition residuals in the ANSYS Fluent console. These residuals are calculated as follows with respect to their immediate previous value only.

Pres-m(n) = [abs (m(n) - m(n-1))]/m(n)

Where:

Pres- m(n) is the printed residual value for a report definition

n is the iteration that is currently printed in the console

m(n) is the value of the monitor at the n^{th} iteration

Therefore:

Pres-m(n +1) = [abs (m(n+1) - m(n))]/m(n+1)

The printed value at the (n+1)th iteration is calculated with respect to the (n)th iteration and the printed value at the (n)th iteration is calculated with respect to the (n-1)th iteration.

28.17. Executing Commands During the Calculation

As described in Monitoring Solution Convergence (p. 1624) and Animating the Solution (p. 1647), respectively, you can report and monitor various quantities (for example, residuals, force coefficients) and create animations of the solution while the solver is performing calculations. ANSYS Fluent also includes a feature that allows you to define your own command(s) to be executed during the calculation at specified intervals. For example, you can ask ANSYS Fluent to perform gradient adaption after a set number of iterations. You will specify a series of text commands or use the GUI to define the steps to be performed.

Important

Note that the **Calculation Activities** task page provides options to perform the following during the calculation:

- save case and data files
- export transient solution files
- export transient particle history data files

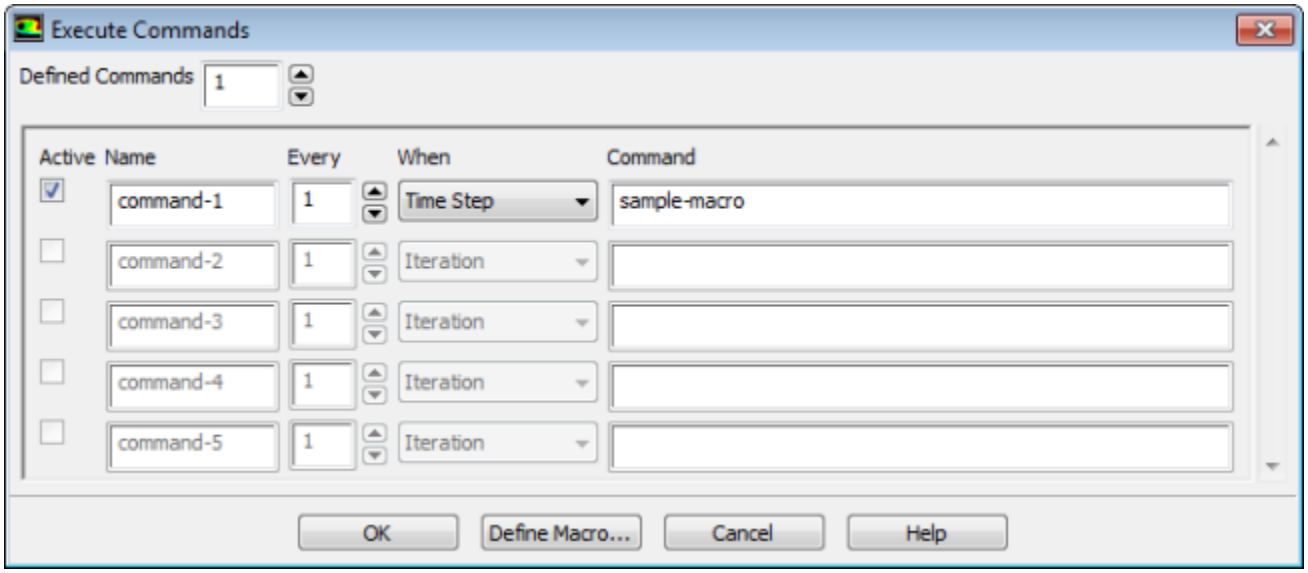
Each of these options has their own dialog box, which should be used rather than executing a command to perform them. See Automatic Saving of Case and Data Files (p. 48) and Exporting Data During a Transient Calculation (p. 86) for details.

You will indicate the command(s) that you want the solver to execute at specified intervals during the calculation using the Execute Commands Dialog Box (p. 2535) (Figure 28.36: The Execute Commands Dialog Box (p. 1639)).

Solution → Calculation Activities → Execute Commands → New...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.36: The Execute Commands Dialog Box



The procedure is as follows:

1. Increase the **Defined Commands** value to the number of commands you want to specify. As this value is increased, additional command entries will become editable. For each command, you will perform the following steps.

2. Enable the **Active** check button next to the command if you want it to be executed during the calculation. You may define multiple commands and choose to use only a subset of them by turning off the check button for those that you do not want to use.

3. Enter a name for the command under the **Name** heading.

4. Indicate how often you want the command to be executed by setting the interval under **Every** and selecting **Iteration** or **Time Step** in the drop-down list below **When**. (**Time Step** is a valid choice only if you are calculating unsteady flow.) For example, to execute the command every 10 iterations, you would enter `10` under **Every** and select **Iteration** under **When**.

Important

If you specify an interval in iterations, be sure to keep the **Reporting Interval** in the Run Calculation Task Page (p. 2537) at its default value of 1.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Define the command by entering a series of text commands in the **Command** field, or by entering the name of a command macro you have defined (or will define) as described in Defining Macros (p. 1640).

Note

Make sure that the text commands you have entered in the **Command** field of the **Execute Commands** dialog box does not exceed 127 characters.

Important

- If the command to be executed involves saving a file, see Saving Files During the Calculation (p. 1642) for important information.
- If the command enables or disables equations, you must disable the convergence check for these equations.

For additional information, see the following sections:
28.17.1. Defining Macros
28.17.2. Saving Files During the Calculation

28.17.1. Defining Macros

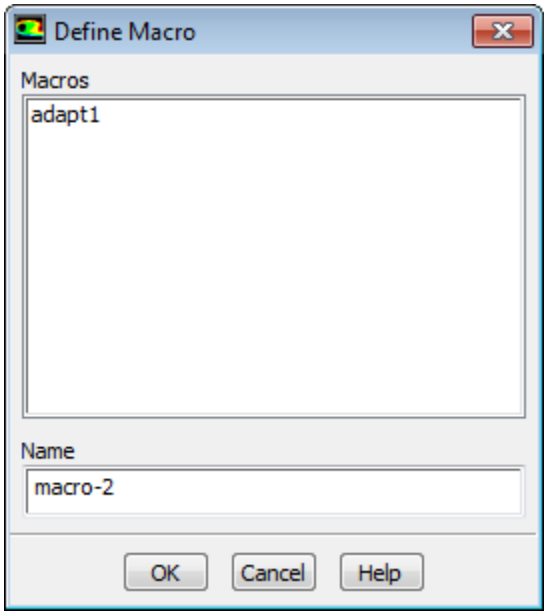
Macros that you define for automatic execution during the calculation can also be used interactively by you during the problem setup or postprocessing. For example, if you define a macro that performs a certain type of adaption after each iteration, you can also use the macro to perform this adaption interactively.

Definition of a macro is accomplished as follows:

1. In the Execute Commands Dialog Box (p. 2535), click the **Define Macro...** button to open the Define Macro Dialog Box (p. 2536) (Figure 28.37: The Define Macro Dialog Box (p. 1641)). Since this is a "modal" dialog box, the solver will not allow you to do anything else until you perform step 2, below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.37: The Define Macro Dialog Box



2. In the **Define Macro** dialog box, specify a **Name** for the macro (for example, `adapt1`) and click **OK**. (The **Define Macro...** button in the **Execute Commands** dialog box will become the **End Macro** button.)

3. Perform the steps that you want the macro to perform. For example, if you want the macro to perform gradient adaption, open the **Gradient Adaption** dialog box, specify the appropriate adaption function and parameters, and click **Adapt** to perform the adaption.

   Important

   If the command to be executed involves saving a file, see Saving Files During the Calculation (p. 1642) for important information.

4. When you have completed the steps you want the macro to perform, click the **End Macro** button in the **Execute Commands** dialog box.

As noted above, once you have defined a macro for execution during the calculation, you can use it at any time. If you defined the macro called `adapt1` to adapt based on pressure gradient, you can simply type `adapt1` in the console to perform this adaption. This macro is independent of any text menus, so you need not move to a different text menu to use it. Macros can be saved to and read from files. To save all macros that are currently defined, use the `file/write-macros` text command. To read all the macros in a macro file, use the `file/read-macros` text command.

Important

A macro, like a journal file, is a simple record/playback function. It will therefore know nothing about the state in which it was recorded or the state in which it is being played back. You must be careful not to change directories while defining a macro. Also, you must be careful that all surfaces, variables, and so on, that are used by the macro have been properly defined when you (or ANSYS Fluent) invoke the macro.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.17.2. Saving Files During the Calculation

If the command to be executed during the calculation involves saving a file, you should include a special character in the file name when you enter it in The Select File Dialog Box (p. 19) so that the solver will know to assign a new name to each file it saves. See Automatic Numbering of Files (p. 43) for details about these special characters for filenames.

Important

Note that the **Calculation Activities** task page provides options to perform the following during the calculation:

- save case and data files
- export transient solution files
- export transient particle history data files

Each of these options has their own dialog box, which should be used rather than the **Execute Commands** dialog box. See Automatic Saving of Case and Data Files (p. 48) and Exporting Data During a Transient Calculation (p. 86) for details.

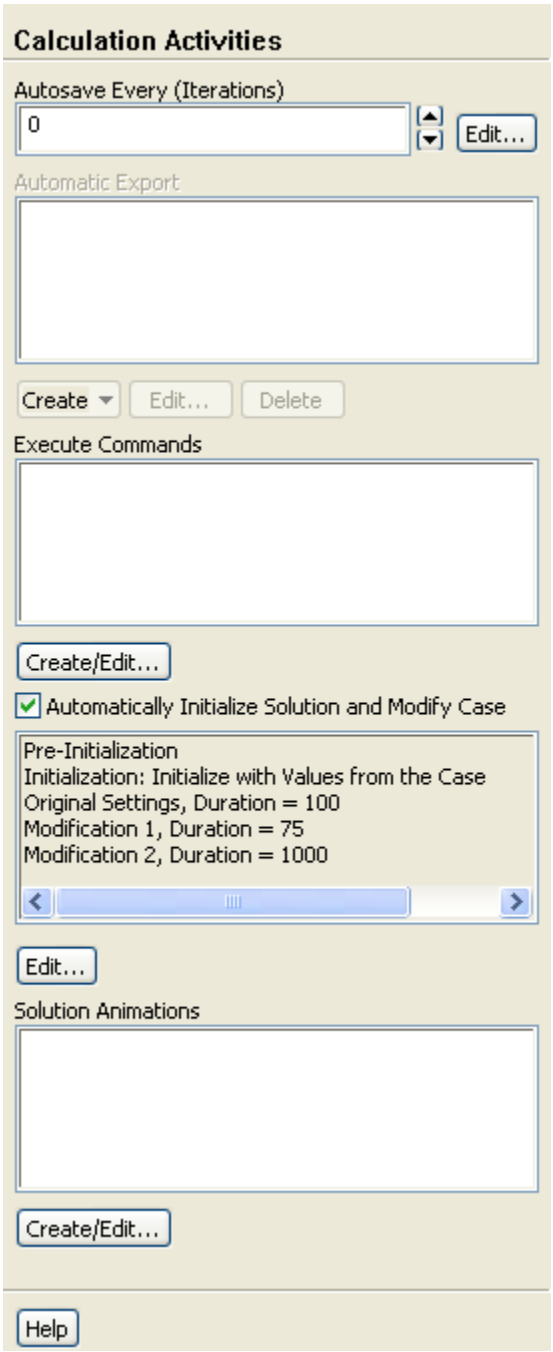
28.18. Automatic Initialization of the Solution and Case Modification

While running a case manually, you can perform certain activities that may facilitate convergence. These actions may take place before initialization, after initialization and/or at other points during the calculation. The process described in this section allows you to enter text commands at each of these times when a case is run from the **Run Calculation** task page, Workbench, or in batch. In the **Calculation Activities** task page, enable **Automatically Initialize Solution and Modify Case** to automatically modify the case.

Solution → Calculation Activities → Automatically Initialize Solution and Modify Case → Edit...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.38: The Automatically Initialize Solution and Modify Case Option



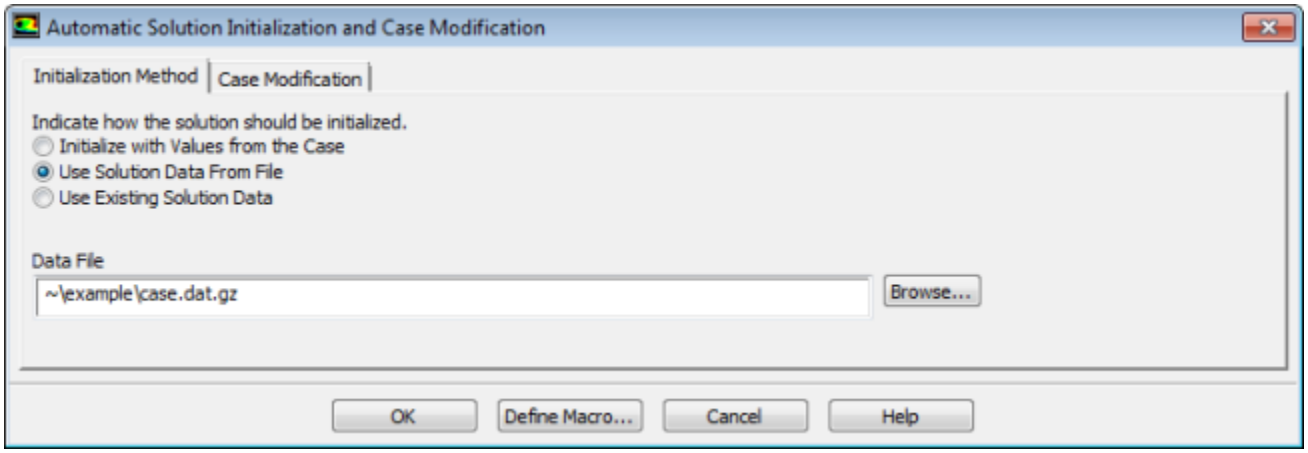
When using this option, you can edit the calculation settings. Note that the original settings always exist and cannot be deleted. The duration of the calculation is defined, so immediately after enabling **Automatically Initialize Solution and Modify Case**, you will notice that the **Calculation Activities** task page reports that the case will be run with the original settings for a single iteration.

You can now control the number of iterations or time steps for the calculation. When **Automatically Initialize Solution and Modify Case** option is disabled, you will have to specify the iterations or time steps using the **Run Calculation** task page.

For an uninitialized case, clicking the **Edit...** button will display the **Automatic Solution Initialization and Case Modification** dialog box, which allows you to specify the initialization method and to modify the case.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.39: The Automatic Solution Initialization and Case Modification Dialog Box



In the **Initialization Method** tab, you can specify four different initialization methods:

Initialize with Values from the Case
uses the values set in the **Solution Initialization** task page.

> **Note**
>
> Hybrid initialization is not performed for this option; even if you have selected **Hybrid Initialization** under the **Solution Initialization** task page.

Use Solution Data from File
requires you to read in a data file containing the desired initialization for this case, as shown in Figure 28.39: The Automatic Solution Initialization and Case Modification Dialog Box (p. 1644).

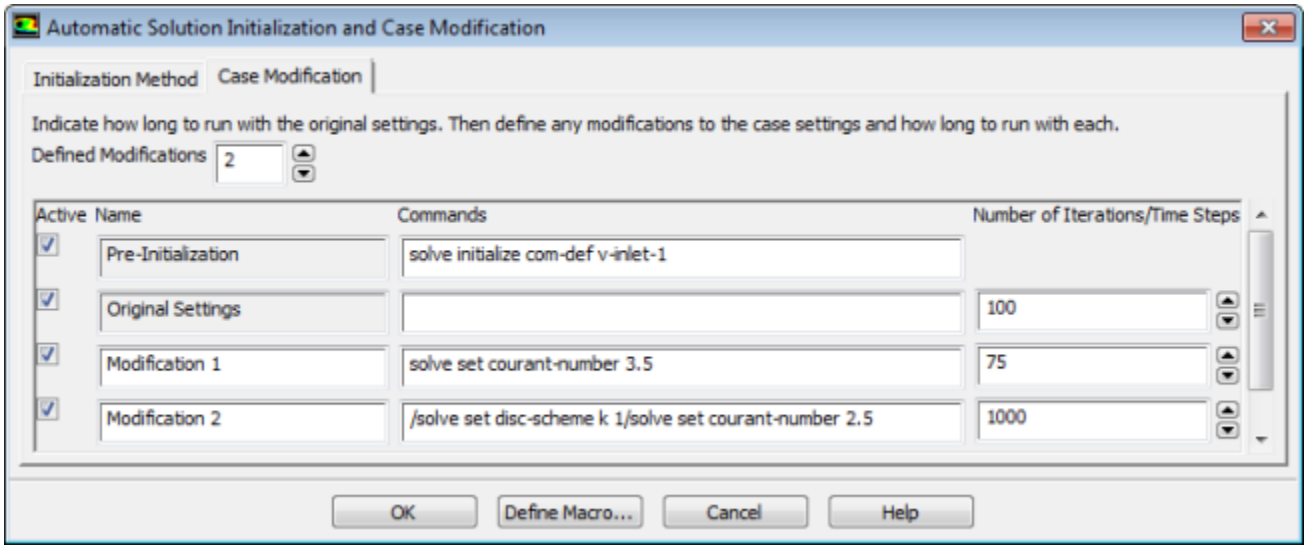
Use Existing Solution Data
is analogous to changing the values in a case and continuing the calculation. However, the iteration counter will be reset to 0 so that the modifications can be applied. Use this method when no solution data exists, similar to the first run.

> **Important**
>
> Whenever the case is initialized, the iteration count is set to 0.

In the **Case Modification** tab, you can indicate how long you would like to run with the original settings, then make any modifications to the case settings.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.40: The Case Modification Tab



If you decide to make no modifications, then the counter in the **Defined Modifications** box will be set to 0. However, you still have the option to specify settings that you would like to apply before initialization, or you can change your original settings. If **Before Initialization** is enabled, then you can type the text commands in the **Commands** field. If **Original Settings** is enabled, you can type text commands in the **Commands** field, and/or specify the **Number of Iterations/Time Steps**. When entering more than one text command in a single **Commands** field, begin each text command with a /, as shown in Figure 28.40: The Case Modification Tab (p. 1645).

Note

Make sure that the text commands you have entered in the **Commands** field does not exceed 127 characters.

Important

Note that some text commands require arguments (for example, numbers, file names, yes/no responses), which are requested through follow-up prompts in the console. When entering such text commands in the **Commands** field of the **Case Modification** tab, be sure to include your responses to the prompts. If you do not include these responses, the associated command will not be executed (unless it is a yes/no question, in which case ANSYS Fluent will use yes by default).

If you decide to run the calculation further with modifications to your case, increase the number of **Defined Modifications** and specify additional commands. The settings shown in Figure 28.40: The Case Modification Tab (p. 1645) result in the following actions:

1. The default initial values for initialization are computed from the velocity-inlet zone **inlet-6**.
2. The iteration count is set to 0 (in all situations) and the solution is initialized.
3. The calculation is run for 100 iterations or until convergence.
4. The Courant number is set to 3.5.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. The calculation continues for 75 iterations or until convergence.

6. The discretization scheme for turbulent kinetic energy is set to second order upwind, and the Courant number is set to 2.5.

7. The calculation continues for 1000 iterations or until convergence.

You can make the above changes sequentially and run your case, or you can specify them all at once. When you have completed making the modifications, click **OK**. A **Question** dialog box may appear, prompting you to take specific actions. For example, if the **Original Settings** field is empty, then you may be notified that the original settings will be lost if the case is saved after the modifications are applied. It will prompt you for a response when asked if you would like to add commands that specify the original settings.

Important

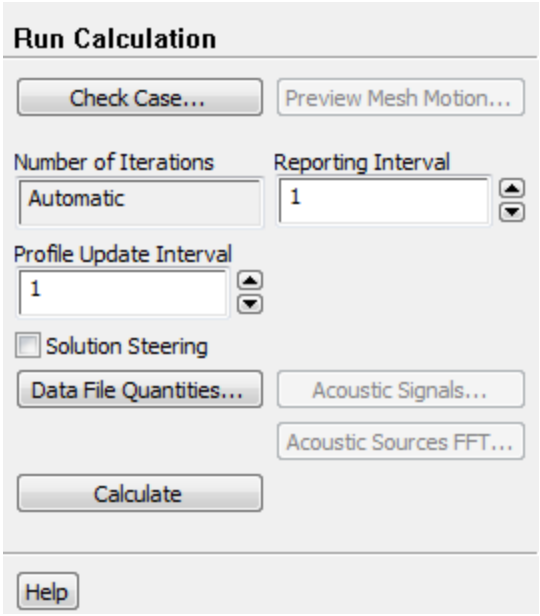
If you specify commands for the **Original Settings**, they will be applied to the case before the first iteration/time step.

Your actions will be summarized in the **Calculation Activities** task page, as shown in Figure 28.38: The Automatically Initialize Solution and Modify Case Option (p. 1643).

If you disable **Automatically Initialize Solution and Modify Case**, the settings will be disabled and retained, but will not be applied to the case.

When **Automatically Initialize the Solution and Modify the Case** is enabled, settings defined in the **Run Calculation** task page will be ignored. Instead, the **Number of Iterations** will be defined as **Automatic**, as shown in Figure 28.41: The Run Calculation Task Page (p. 1646).

Figure 28.41: The Run Calculation Task Page



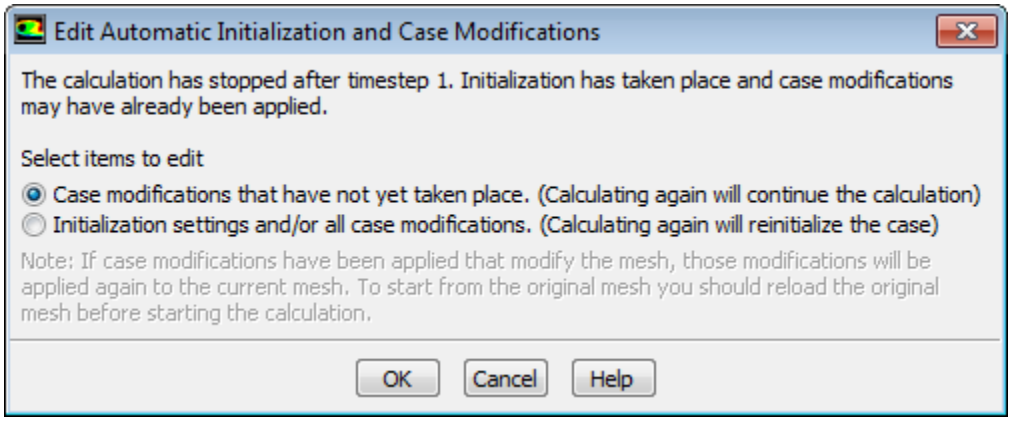
For additional information, see the following sections:
28.18.1. Altering the Solution Initialization and Case Modification after Calculating

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.18.1. Altering the Solution Initialization and Case Modification after Calculating

If you decide to edit the solution initialization and case modification settings and one or more iterations have been calculated, then clicking the **Edit...** button for the **Automatically Initialize Solution and Modify Case** option will open the **Edit Automatic Initialization and Case Modifications** dialog box, as shown in Figure 28.42: The Edit Automatic Initialization and Case Modifications Dialog Box (p. 1647).

Figure 28.42: The Edit Automatic Initialization and Case Modifications Dialog Box



If you select the first option, the initialization controls and modifications that have already taken place are disabled, therefore you can edit the case modifications that have yet to take place.

If you select the second option, all controls in the **Automatic Solution Initialization and Case Modification** dialog box are enabled and therefore, you can modify any of the settings.

28.19. Animating the Solution

During the calculation, you can have Fluent create an animation of mesh (primarily for dynamic mesh cases), contours, vectors, pathlines, particle tracks, scenes, XY plots, report plots, or residuals. Before you begin the calculation, you will specify and display the variables and types of plots you want to animate, and how often you want plots to be saved. At the specified intervals, Fluent will display the requested plots, and store each one. When the calculation is complete, you can play back the animation sequence, modify the view (for mesh, contour, vector, pathline, and particle tracks plots), if desired, and save the animation to a series of picture files or an MPEG file.

Instructions for defining a solution animation sequence are provided in Creating an Animation Definition (p. 1647). Playing an Animation Sequence (p. 1650) describes how to play back and save the animations.

For additional information, see the following sections:

28.19.1. Creating an Animation Definition
28.19.2. Playing an Animation Sequence
28.19.3. Saving an Animation Sequence
28.19.4. Reading an Animation Sequence

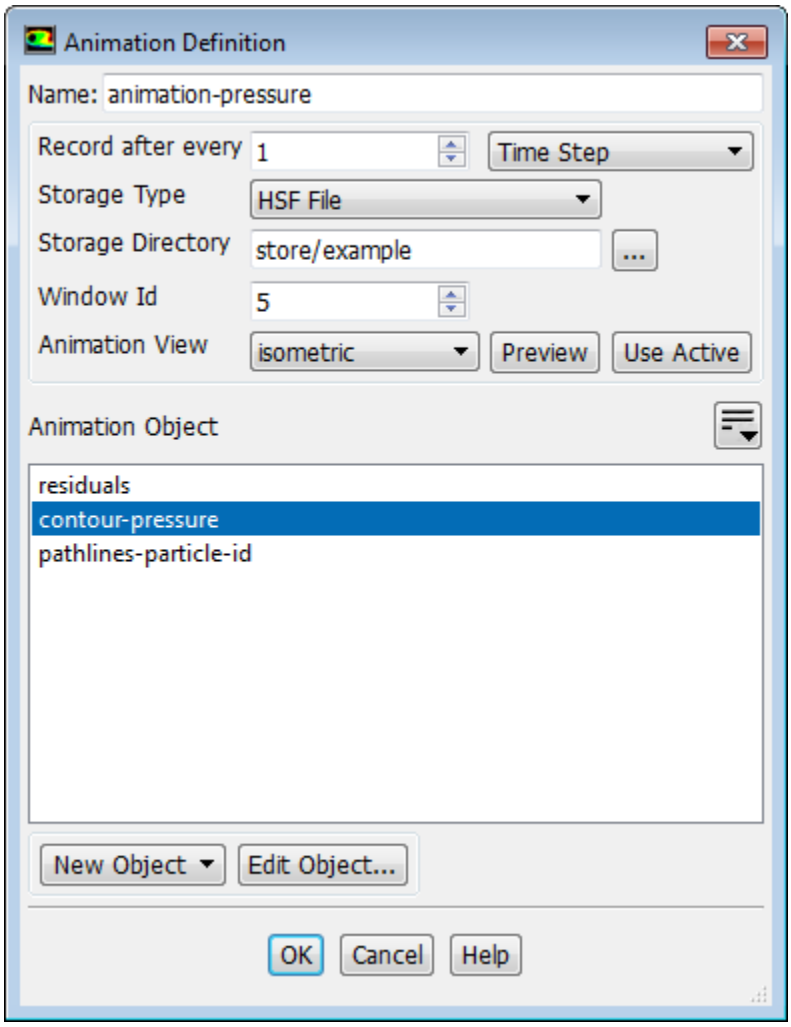
28.19.1. Creating an Animation Definition

You can use Figure 28.43: The Animation Definition Dialog Box (p. 1648) to specify the animation object and the frequency that it will be captured for animating.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solving → Activities → Create → Solution Animations...

Figure 28.43: The Animation Definition Dialog Box



The procedure is as follows:

1. (Optional) Enter a **Name** for the animation definition.

2. Indicate how often you want to create a new frame in the animation by specifying a frequency in **Record after every** and selecting **Iteration** or **Time Step**.

3. Specify how you want Fluent to save the animation frames by selecting **In Memory**, **PPM Image** (2D image file), or **HSF File** (3D image file) from the **Storage Type** drop-down.

Important

The advantage to saving the animation sequence using the **HSF File** option is that these files are highly compressible and can be viewed and interacted with using a "HOOPS Viewer" application on iOS and Android equipped devices. The "HOOPS Viewer" application can be downloaded to iOS and Android equipped devices from the "App Store" and "Google Play" respectively.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

An advantage to saving the animation sequence using the **PPM Image** option is that you can use the separate pixmap image files for the creation of a single GIF file. GIF file creation can be done quickly with graphics tools provided by other third-party graphics packages such as ImageMagick, that is, `animate` or `convert`. For example, if you save the PPM files starting with the string `sequence-2`, and you are using the ImageMagick software, you can use the `convert` command with the `-adjoin` option to create a single GIF file out of the sequence using the following command.

```
convert -adjoin sequence-2_00*.ppm sequence2.gif
```

4. Choose the **Storage Directory** where Fluent save the animation frames.

5. (Optional) Specify the graphics **Window ID** where the animation frames will be displayed during the calculation.

6. (Optional) Specify the **Animation View**, which determines the orientation of the object for the animation frames captured during the calculation.

   *You can only specify the view once an **Animation Object** is selected*. Clicking **Preview** displays the selected object in the graphics window specified by **Window ID**, with the view shown in the drop-down. If your desired view is not available in the drop-down, you can preview the object, change its orientation in the graphics window, then click **Use Active** to save the current graphics window view and use it for this animation definition.

7. Select the graphics object you want to animate from the **Animation Object** list. If you want to animate an object that is not listed, you can create a **New Object**.

   - **Mesh...** opens the Mesh Display Dialog Box (p. 2163).
   - **Contours...** opens the Contours Dialog Box (p. 2676).
   - **Vectors...** opens the Vectors Dialog Box (p. 2831).
   - **Pathlines...** opens the Pathlines Dialog Box (p. 2771).
   - **Particle Tracks...** opens the Particle Tracks Dialog Box (p. 2761).
   - **Scene...** opens Figure 31.26: The Scene Dialog Box (p. 1781).
   - **XY Plot...** opens the Solution XY Plot Dialog Box (p. 2595).
   - **Report Plot...** opens the New Report Plot Dialog Box (p. 2755).

8. Click **OK** to create the animation definition.

28.19.1.1. Guidelines for Creating an Animation Definition

If you are defining an animation sequence containing mesh, contour, or vector displays, note the following when you are defining the display:

- If you want to include lighting effects in the animation frames, be sure to define the lights before you begin the calculation. See Adding Lights (p. 1796) for information about adding lights to the display.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you want to maintain a constant range of colors in a contour or vector display, you can specify a range explicitly by turning off the **Auto Range** option in the **Contours** or **Vectors** dialog box. See Specifying the Range of Magnitudes Displayed (p. 1759) or Specifying the Range of Magnitudes Displayed (p. 1768) for details.

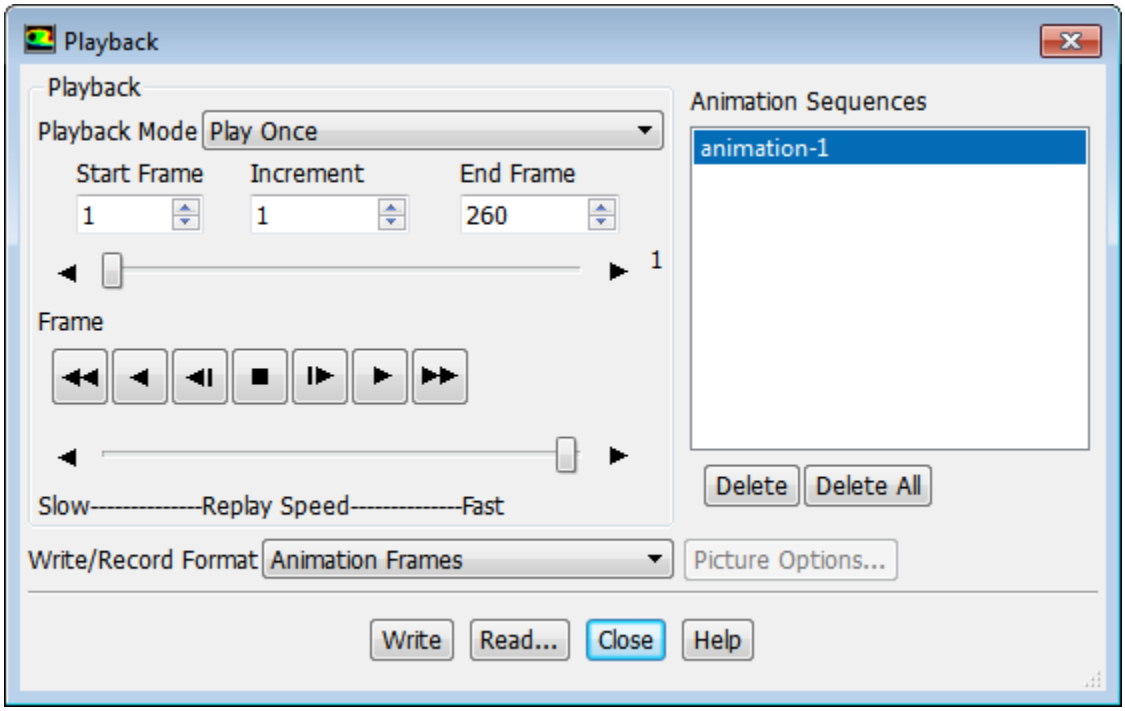
- Advanced scene manipulations that are specified using the **Scene Description** dialog box will *not* be included in the animation sequence frames. View modifications such as mirroring across a symmetry plan *will* be included.

28.19.2. Playing an Animation Sequence

Once you have defined a sequence (as described in Creating an Animation Definition (p. 1647)) and performed a calculation, or read in a previously created animation sequence (as described in Reading an Animation Sequence (p. 1654)), you can play back the sequence using the Playback Dialog Box (p. 2571) (Figure 28.44: The Playback Dialog Box (p. 1650)).

Results → Animations → Solution Animation Playback Edit...

Figure 28.44: The Playback Dialog Box



Under **Animation Sequences** in the **Playback** dialog box, select the sequence you want to play in the **Sequences** list. To play the animation once through from start to finish, click the "play" button under the **Playback** heading. (The buttons function in a way similar to those on a standard video cassette player. "Play" is the second button from the right—a single triangle pointing to the right.) To play the animation backwards once, click the "play reverse" button (the second from the left—a single triangle point to the left). As the animation plays, the **Frame** scale shows the number of the frame that is currently displayed, as well as its relative position in the entire animation. If, instead of playing the complete animation sequence, you want to jump to a particular frame, move the **Frame** slider bar to the desired

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

frame number, and the frame corresponding to the new frame number will be displayed in the graphics window.

Important

For smoother animations, enable the **Double Buffering** option in the Display Options Dialog Box (p. 2573) (see Modifying the Rendering Options (p. 1798)). This will reduce screen flicker during graphics updates.

Additional options for playing back animations are described below.

28.19.2.1. Modifying the View

If you want to replay the animation sequence with a different view of the scene, you can use your mouse to modify (for example, translate, rotate, zoom) it in the graphics window where the animation is displayed. Note that any changes you make to the view for an animation sequence will be lost when you select a new sequence (or reselect the current sequence) in the **Sequences** list.

28.19.2.2. Modifying the Playback Speed

Different computers will play the animation sequence at different speeds, depending on the complexity of the scene and the type of hardware used for graphics. You may want to slow down the playback speed for optimal viewing. Move the **Replay Speed** slider bar to the left to reduce the playback speed (and to the right to increase it).

28.19.2.3. Playing Back an Excerpt

You may sometimes want to play only one portion of a long animation sequence. To do this, you can modify the **Start Frame** and the **End Frame** under the **Playback** heading. For example, if your animation contains 50 frames, but you want to play only frames 20 to 35, you can set **Start Frame** to 20 and **End Frame** to 35. When you play the animation, it will start at frame 20 and finish at frame 35.

28.19.2.4. "Fast-Forwarding" the Animation

You can "fast-forward" or "fast-reverse" the animation by skipping some of the frames during playback. To fast-forward the animation, you will need to set the **Increment** and click the fast-forward button (the last button on the right—two triangles pointing to the right). If, for example, your **Start Frame** is 1, your **End Frame** is 15, and your **Increment** is 2, when you click the fast-forward button, the animation will show frames 1, 3, 5, 7, 9, 11, 13 and 15. Clicking on the fast-reverse button (the first button on the left—two triangles pointing to the left) will show frames 15, 13, 11,...1.

28.19.2.5. Continuous Animation

If you want the playback of the animation to repeat continuously, there are two options available. To continuously play the animation from beginning to end (or from end to beginning, if you use one of the reverse play buttons), select **Auto Repeat** in the **Playback Mode** drop-down list. To play the animation back and forth continuously, reversing the playback direction each time, select **Auto Reverse** in the **Playback Mode** drop-down list.

To turn off the continuous playback, select **Play Once** in the **Playback Mode** list. This is the default setting.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.19.2.6. Stopping the Animation

To stop the animation during playback, click the "stop" button (the square in the middle of the playback control buttons). If your animation contains very complicated scenes, there may be a slight delay before the animation stops.

28.19.2.7. Advancing the Animation Frame by Frame

To advance the animation manually frame by frame, use the third button from the right (a vertical bar with a triangle pointing to the right). Each time you click this button, the next frame will be displayed in the graphics window. To reverse the animation frame by frame, use the third button from the left (a left-pointing triangle with a vertical bar). Frame-by-frame playback allows you to freeze the animation at points that are of particular interest.

28.19.2.8. Deleting an Animation Sequence

If you want to remove one of the sequences that you have created or read in, select it in the **Sequences** list and click the **Delete** button. If you want to delete all sequences, click the **Delete All** button.

Important

Note that if you delete a sequence that has not yet been saved to disk (that is, if you selected **In Memory** under **Storage Type** in the **Animation Sequence** dialog box), it will be removed from memory permanently. If you want to keep any animation sequences that are stored only in memory, you should be sure to save them (as described in Saving an Animation Sequence (p. 1652)) before you delete them from the **Sequences** list or exit ANSYS Fluent.

28.19.3. Saving an Animation Sequence

Once you have created an animation sequence, you can save it in any of the following formats:

- Solution animation file containing the ANSYS Fluent metafiles
- Picture files, each containing a frame of the animation sequence
- MPEG file containing each frame of the animation sequence

Note that, if you are saving picture files or an MPEG file, you can modify the view (for example, translate, rotate, zoom) in the graphics window where the animation is displayed, and save the modified view instead of the original view.

28.19.3.1. Solution Animation File

If you selected **HSF File** or **PPM Image** for **Storage Type** in Figure 28.43: The Animation Definition Dialog Box (p. 1648), then Fluent will save the solution animation file for you automatically. It will be saved in the specified **Storage Directory**, and its name will be the **Name** you specified for the sequence, with a `.cxa` extension (for example, `pressure-contour.cxa`). In addition to the `.cxa` file, Fluent will also save image files with `.hsf` or `.ppm` extensions for each frame (for example, `pressure-contour_0002.hsf`). The `.cxa` file contains a list of the associated `.hsf` or `.ppm` files, and tells Fluent the order in which to display them.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you selected **In Memory** under **Storage Type**, then the solution animation file (`.cxa`) and the associated files (`.hmf`) will be lost when you exit from ANSYS Fluent, unless you save them as described below.

You can save the animation sequence to a file that can be read back into ANSYS Fluent (see Reading an Animation Sequence (p. 1654)) when you want to replay the animation. As noted in Reading an Animation Sequence (p. 1654), the solution animation file can be used for playback in Fluent independent of the case and data files that were used to generate it.

To save a solution animation file (and the associated files), select **Animation Frames** in the **Write/Record Format** drop-down list in the **Playback** dialog box, and click the **Write** button. ANSYS Fluent will save a `.cxa` file, as well as a `.hmf` file for each frame of the animation sequence. The filename for the `.cxa` file will be the specified sequence **Name** (for example, `pressure-contour.cxa`), and the file names for the metafiles will consist of the specified sequence **Name** followed by a frame number (for example, `pressure-contour_0002.hmf`). All of the files (`.cxa` and `.hmf`) will be saved in the current working directory.

28.19.3.2. Picture File

You can also generate a picture file for each frame in the animation sequence. This feature allows you to save your sequence frames to picture files used by an external animation program such as ImageMagick. As noted above, you can modify the view in the graphics window before you save the picture files.

To save the animation as a series of picture files, follow these steps:

1. Select **Picture Files** in the **Write/Record Format** drop-down list in the **Playback** dialog box.

2. If necessary, click the **Picture Options...** button to open the Save Picture Dialog Box (p. 2568) and set the appropriate parameters for saving the picture files. (If you are saving picture files for use with ImageMagick, for example, you may want to select the window dump format. See Window Dumps (Linux Systems Only) (p. 106) **Apply** in the **Save Picture** dialog box to save your modified settings.

   Important

   Do not click the **Save...** button in the **Save Picture** dialog box. You will save the picture files from the **Playback** dialog box in the next step.

3. In the **Playback** dialog box, click the **Write** button. ANSYS Fluent will replay the animation, saving each frame to a separate file. The filenames will consist of the specified sequence **Name** followed by an animation sequence and a frame number (for example, `pressure-contour_1_0002.ps`), and they will all be saved in the current working directory.

28.19.3.3. MPEG File

It is also possible to save all of the frames of the animation sequence in an MPEG file, which can be viewed using an MPEG decoder such as **mpeg_play**. Saving the entire animation to an MPEG file will require less disk space than storing individual window dump files (using the picture method), but the MPEG file will yield lower-quality images.

As noted above, you can modify the view in the graphics window before you save the MPEG file.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To save the animation to an MPEG file, follow these steps:

1. Select **MPEG** in the **Write/Record Format** drop-down list in the **Playback** dialog box.

2. Click the **Write** button.

   ANSYS Fluent will replay the animation and save each frame to a separate scratch file, and then it will combine all the files into a single MPEG file. The name of the MPEG file will be the specified sequence **Name** with an `.mpg` extension (for example, `pressure-contour.mpg`), and it will be saved in the current working directory.

28.19.4. Reading an Animation Sequence

If you have saved an animation sequence to a solution animation file (as described in Saving an Animation Sequence (p. 1652)), you can read that file back in at a later time (or in a different session) and play the animation. Note that you can read a solution animation file into any ANSYS Fluent session; you do not need to read in the corresponding case and data files. In fact, you do not need to read in any case and data files at all before you read a solution animation file into ANSYS Fluent.

To read a solution animation file, click the **Read...** button in the Playback Dialog Box (p. 2571). In The Select File Dialog Box (p. 19), specify the name of the file to be read.

28.20. Checking Your Case Setup

After you have set up your case, and prior to solving it, you can check your case setup using the Case Check Dialog Box (p. 2543) (Figure 28.45: The Case Check Dialog Box (p. 1655)). This function provides you with guidance and best practices when choosing case parameters and models. Your case will be checked for compliance in the mesh, models, boundary and cell zone conditions, material properties, and solver categories. Established rules will be available for each category, with recommended changes to your current settings. At your discretion, you may elect to apply the recommendations, or keep your current settings.

To access the Case Check Dialog Box (p. 2543) (Figure 28.45: The Case Check Dialog Box (p. 1655)), go to

Solution → **Run Calculation** → **Check Case...**

If there are no problems with your case setup, then an information dialog box (Figure 28.46: The Information Dialog Box (p. 1655)) will appear stating that no recommendations need to be made at this time, otherwise, the Case Check Dialog Box (p. 2543) will open.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.45: The Case Check Dialog Box

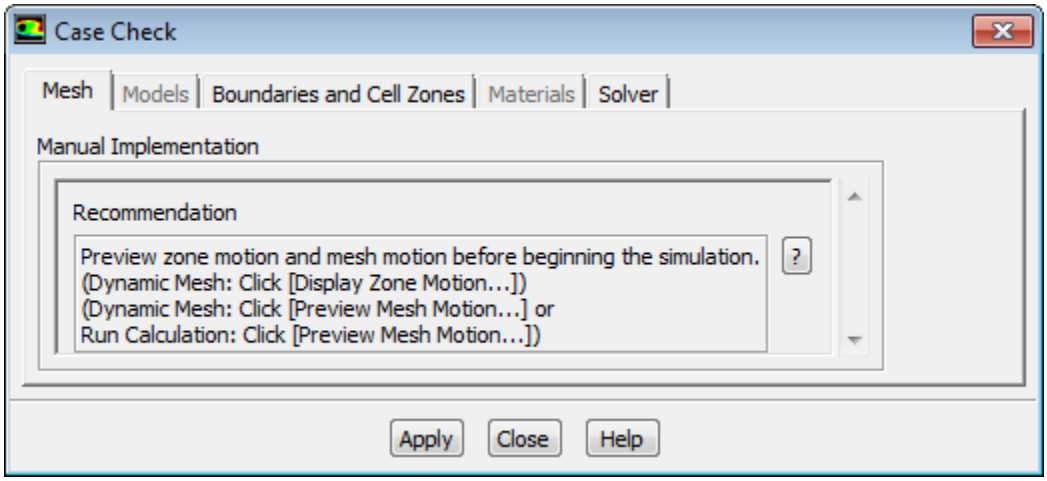
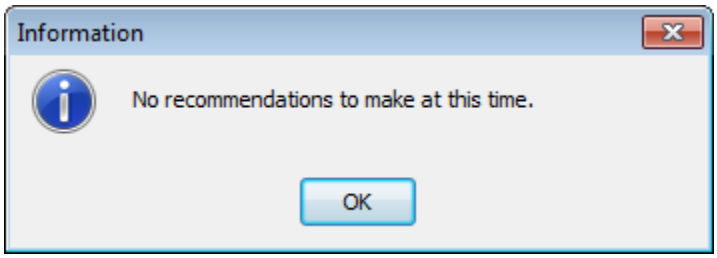


Figure 28.46: The Information Dialog Box



In the **Case Check** dialog box, each of the tabs **Mesh**, **Models**, **Boundaries and Cell Zones**, **Materials**, and **Solver** may contain recommendations. For each of the tabs that are enabled, best practices will be listed.

In some cases, the dialog box will be split based on the method that the recommendation is applied. There are two ways you can apply the listed recommendations:

Automatic Implementation
ANSYS Fluent applies the change for you.

Manual Implementation
You will manually change your case settings.

For additional information, see the following sections:
28.20.1. Automatic Implementation
28.20.2. Manual Implementation

28.20.1. Automatic Implementation

To the left of each of the recommendations listed under **Automatic Implementation** (for example, Figure 28.48: The Models Tab in the Case Check Dialog Box (p. 1659)), there is an enabled **Apply** check box. An enabled check box will result in ANSYS Fluent applying the change to your case automatically. If there are some recommendations that you do not want ANSYS Fluent to implement automatically, then click the **Apply** check box to toggle off and disable the implementation of a particular recommendation. After going through all the tabs and determining which rules you want applied automatically, click the **Apply** button at the bottom of the dialog box. Changes to your settings will be applied to all

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

recommendations throughout the dialog box with an enabled **Apply** check box. ANSYS Fluent will print a message in the console notifying you that the applied recommendation has been implemented.

ANSYS Fluent will ask you if you want to save the case before proceeding to the next step. If you choose **Yes**, The Select File Dialog Box (p. 19) will open allowing you to save your case with the new settings. If you select **No**, all the changes made to the case file will be lost once you exit ANSYS Fluent.

28.20.2. Manual Implementation

For recommendations that are listed under **Manual Implementation**, ANSYS Fluent cannot apply the changes for you. Therefore, if you opt to make a change to your current settings, based on the listed recommendations, then you will need to manually make the changes by opening the affected dialog boxes or task pages and applying what was recommended.

To the right of the recommendations is a **?**, which essentially acts as a help button, leading you to related documentation on the specific topic.

At the bottom of each recommendation, there is a path that will guide you to the dialog box or task page where you can make the changes. For example, in the **Mesh** tab, you will see the following recommendation:

```
Check your mesh.
(General: Click [Check])
```

To perform the action, highlight **General** in the tree, then click the **Check** button in the **Mesh** group box. You will see a path for each recommendation, in each of the tabs.

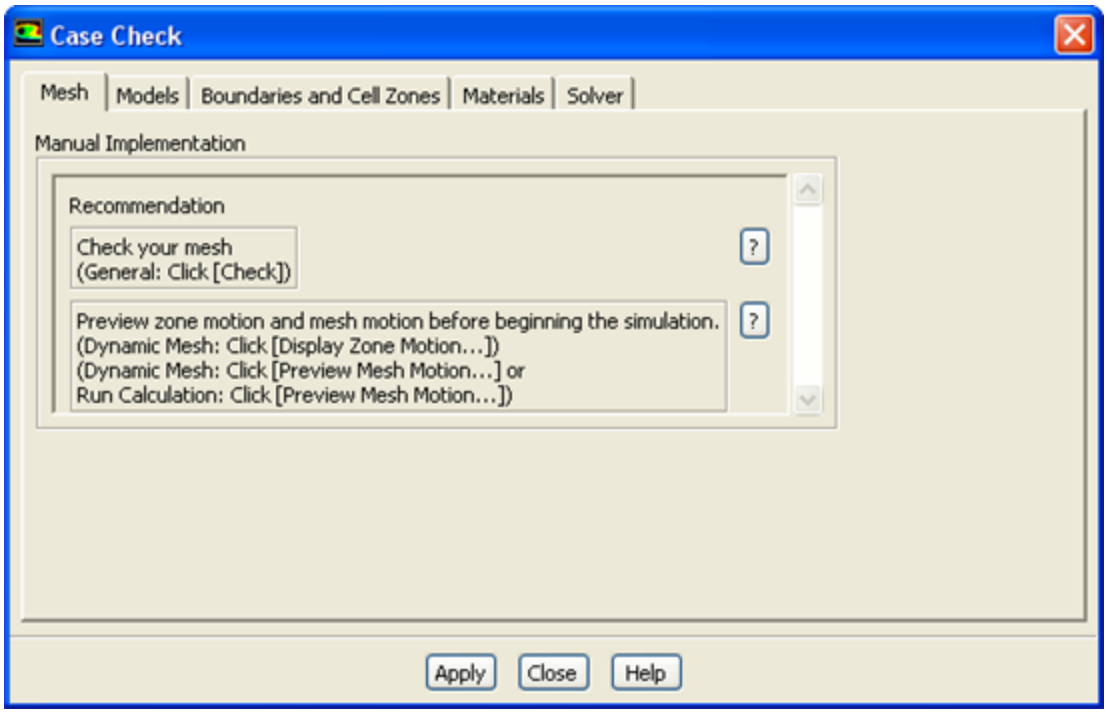
Each of the case check rules are described in the following sections:

28.20.2.1. Checking the Mesh
28.20.2.2. Checking Model Selections
28.20.2.3. Checking Boundary and Cell Zone Conditions
28.20.2.4. Checking Material Properties
28.20.2.5. Checking the Solver Settings

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.20.2.1. Checking the Mesh

Figure 28.47: The Mesh Tab in the Case Check Dialog Box



The following recommendations appear under the **Mesh** tab (Figure 28.47: The Mesh Tab in the Case Check Dialog Box (p. 1657)):

- **Check your mesh.**

  If you have not already checked your mesh, it is best practice that you do so immediately after reading in your mesh, or after any mesh modification. To check your mesh go to

  Setup → General → Check

  Checking the mesh will help you detect any mesh trouble before you get started with your problem setup. You can learn more about the information obtained when checking your mesh, by going to Checking the Mesh (p. 192).

- **Improve the mesh quality before proceeding with your simulation. The maximum cell skewness is greater than 0.98.**

  Check the quality of your mesh immediately after reading in your mesh, or after any mesh modification. The quality of the mesh plays a significant role in the accuracy and stability of the numerical computation. You can learn more about the quality of your mesh by going to Mesh Quality (p. 131).

  Setup → General → Report Quality

- **Preview zone motion and mesh motion before beginning the simulation.**

  After setting up your case using the **Dynamic Mesh** model, it is worth while to preview your mesh prior to running your simulation. You can preview **Zone Motion** by going to

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → Dynamic Mesh → Dynamic Mesh → Display Zone Motion...

To preview the **Mesh Motion**, go to

Setup → Dynamic Mesh → Dynamic Mesh → Preview Mesh Motion...

or

Solution → Run Calculation → Preview Mesh Motion... **Solution → Run Calculation → Preview Mesh Motion...**

It is important that you preview zone motion first and then mesh motion. Zone motion shows the motion of all dynamic zones with the prescribed rigid body motion, using the graphics library. It is a very fast process and does not alter the mesh. Previewing the zone motion will show you if the motion is set up properly (for example zones moving in the wrong direction or rotating about the wrong center). It is much more difficult to detect these problems with mesh motion because small time steps are performed and no continuous animation is shown.

Mesh motion should always be done for dynamic mesh cases with prescribed motion. Mesh motion will only show the validity of the mesh during the simulation. Mesh deformation and dynamic zones without rigid body motion will be considered during a mesh motion preview.

Both the **Mesh Motion** and **Zone Motion** dialog boxes will have a **Preview** button that will allow you to view the mesh or zone motion prior to running your case. You can obtain more information on mesh motion and zone motion by going to Previewing the Dynamic Mesh (p. 742).

- **Translate the mesh for axisymmetric geometry containing nodes below the x-axis.**

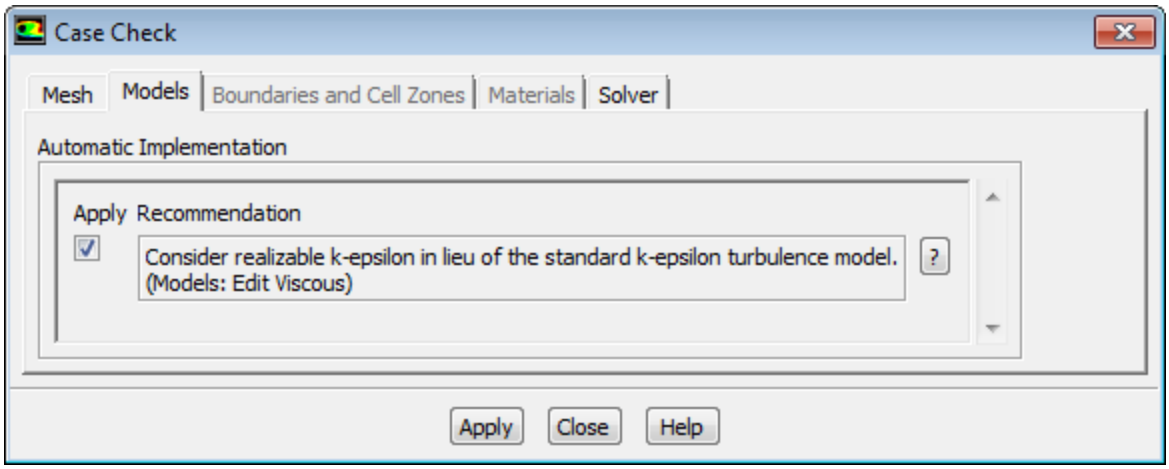
  If either **Axisymmetric** or **Axisymmetric Swirl** is specified in the **General** task page and there are mesh nodes that fall below the x-axis, then it is recommended that you translate the mesh. Nodes below the x axis are forbidden for axisymmetric cases, since the axisymmetric cell volumes are created by rotating the 2D cell volume about the x axis; therefore nodes below the x axis would create negative volumes. To find out if there are any nodes that lie below the x-axis, perform a mesh check (Checking the Mesh (p. 192)). For information on translating the mesh, see Translating the Mesh (p. 228). To access the **Mesh Translate** dialog box, go to

  Setting Up Domain → Mesh → Transform → Translate...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.20.2.2. Checking Model Selections

Figure 28.48: The Models Tab in the Case Check Dialog Box



The following recommendations appear under the **Models** tab (Figure 28.48: The Models Tab in the Case Check Dialog Box (p. 1659)):

- **Consider realizable k-epsilon in lieu of the standard k-epsilon turbulence model.**

  The realizable k- ε model is a more recent development of the standard k- ε model and differs from it in that the realizable k- ε model contains a new formulation for the turbulent viscosity, as well as a new transport equation for the dissipation rate, ε, derived from an exact equation for the transport of the mean-square vorticity fluctuation.

  realizable k- ε model means that the model satisfies certain mathematical constraints on the Reynolds stresses, consistent with the physics of turbulent flows. For more information on the standard k-ε model and the realizable k- ε model, visit Standard k-ε Model and Realizable k-ε Model (in the Theory Guide), respectively.

  Setup → Models → Viscous → Edit...

  For information on all k- ε model options, go to Standard, RNG, and Realizable k-ε Models in the Theory Guide.

- **Disable DO/Energy coupling if the optical thickness is less than 10.**

  DO/Energy coupling should only be used when the optical thickness is greater than 10. Refer to Energy Coupling and the DO Model in the Theory Guide for more information.

  Setup → Models → Radiation → Edit...

- **Verify that the temperature specified for boundary zones that do not participate in the view factor calculation is appropriate.**

  When using the S2S radiation model, make sure that you set the temperature for boundaries that do not participate in the view factor calculation to an appropriate value. In most cases the appropriate value is the ambient temperature, which by default is assumed to be 300 K. See Specifying Boundary Zone Participation (p. 858) for more information.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → Models → Radiation → Edit...

- **Change the under-relaxation factor for the mixing plane model to 1.0.**

  If you have created a mixing plane, set the **Under-Relaxation** in the **Mixing Plane** dialog box to 1. Look under Global Parameters in Setting Up the Mixing Plane Model (p. 619) for information about the mixing plane under-relaxation.

  Setting Up Domain → Mesh Models → Mixing Planes...

- **Enable the smoothing option for dynamic mesh simulations when remeshing.**

  When your case involves the use of dynamic meshes and remeshing is enabled, then it is recommended that you also perform smoothing on the mesh. For a complete discussion of smoothing and remeshing, see Setting Dynamic Mesh Modeling Parameters (p. 643).

  Setup → Dynamic Mesh → Dynamic Mesh

- **Disable species inlet diffusion for laminar flow with species transport.**

  By default, ANSYS Fluent includes the diffusion flux of species at inlets. In some cases involving species transport and laminar flow, it is recommended that the **Inlet Diffusion** option in the **Species Model** dialog box is disabled. For example,

  – If you want to include only the convective transport of species through the inlets of your domain.

  – If at one of the inlets, the convective flux is very small, resulting in mass loss by diffusion through the inlet.

  Setup → Models → Species → Edit...

  For more information about diffusion at inlets, go to Defining Cell Zone and Boundary Conditions for Species (p. 990).

- **Include turbulence interaction for the NOx model.**

  When running thermal NOx simulations and your flow is turbulent, then be sure to set the NOx **Turbulence Interaction Mode**.

  Setup → Models → NOx → Edit...

  In turbulent combustion calculations, ANSYS Fluent solves the density-weighted time-averaged Navier-Stokes equations for temperature, velocity, and species concentrations or mean mixture fraction and variance. Methods of modeling the mean turbulent reaction rate can be based on either moment methods or probability density function (PDF) techniques. ANSYS Fluent uses the PDF approach.

  To learn about how this feature is set up, go to Setting Turbulence Parameters (p. 1166).

- **Consider using the default Schnerr-Sauer or the Zwart-Gerber-Belamri cavitation model.**

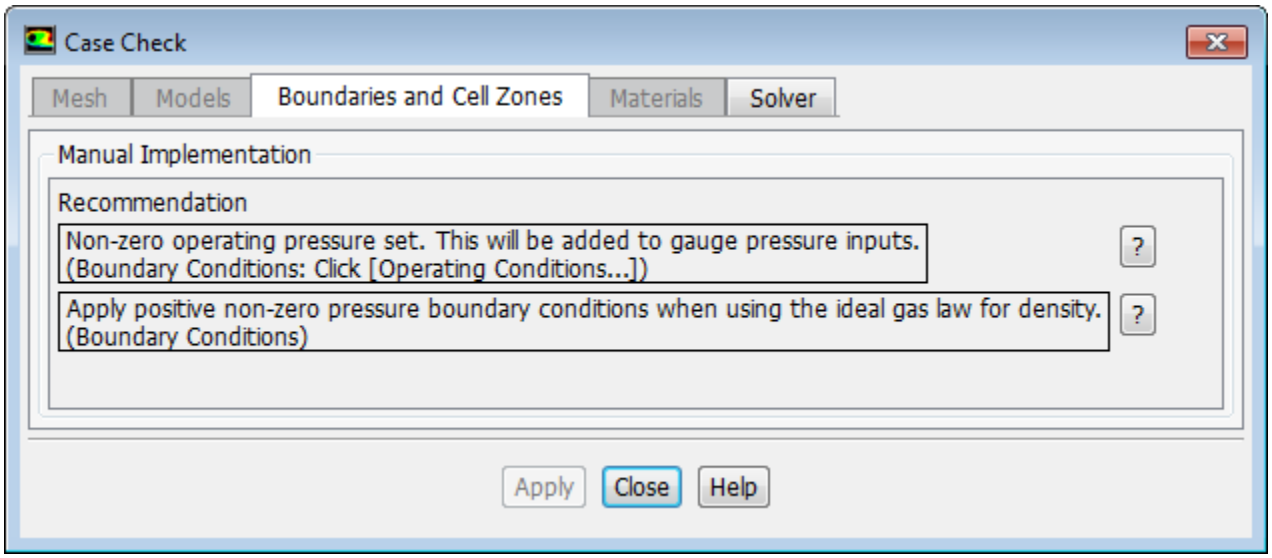
  When using the mixture multiphase model with the Singhal et al. cavitation model enabled, consider changing it to either the Schnerr-Sauer or the Zwart-Gerber-Belamri cavitation model. Refer to Cavitation Models in the Theory Guide for more information.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → Models → Phases → Interaction...

28.20.2.3. Checking Boundary and Cell Zone Conditions

Figure 28.49: The Boundaries and Cell Zones Tab in the Case Check Dialog Box



The following recommendations appear under the **Boundaries and Cell Zones** tab (Figure 28.49: The Boundaries and Cell Zones Tab in the Case Check Dialog Box (p. 1661)):

- **Apply an axis boundary on the centerline (x-axis).**

  For geometry that is axisymmetric or axisymmetric swirl (as set in the **General** task page), the centerline (x-axis) boundary type should be set to **axis**. See Axis Boundary Conditions (p. 389).

  Boundary Conditions

- **Change outlet boundary conditions. A combination of pressure and outflow boundaries is not compatible.**

  Outflow boundary conditions in ANSYS Fluent are used to model flow exits where the details of the flow velocity and pressure are not known prior to solution of the flow problem. One of the limitations when using outflow boundary conditions is that outflow boundary conditions are not compatible with pressure inlets. Therefore, it is recommended that you use velocity or mass flow inlets instead of pressure inlets when used in combination with outflow boundaries. See Outflow Boundary Conditions (p. 350) for a list of limitations that exist with outflow boundaries.

  Setup → Boundary Conditions

- **Change outlet boundary conditions. Outflow boundary conditions are not compatible with the ideal gas law for density.**

  Outflow boundaries cannot be used if you are modeling unsteady flows with varying density, even if the flow is incompressible. See Outflow Boundary Conditions (p. 350) for more limitations that exist with outflow boundaries.

  Setup → Boundary Conditions

- **Non-zero operating pressure set. This will be added to gauge pressure inputs.**

  For cases that have density specified as the ideal gas law, and the operating pressure is greater than zero, the operating pressure will be added to the gauge pressure to yield the absolute pressure. For more information, see Density Inputs for the Ideal Gas Law for Compressible Flows (p. 485) and Operating Pressure, Gauge Pressure, and Absolute Pressure (p. 531).

  Setup → Boundary Conditions → Operating Conditions...

- **Apply positive non-zero pressure boundary conditions when using the ideal gas law for density.**

  In compressible flows , isentropic relations for an ideal gas are applied to relate total pressure, static pressure, and velocity at a pressure inlet boundary. Your input of total pressure, p_0', at the inlet and the static pressure, p_s', in the adjacent fluid cell are related, as described in Equation 6.83 (p. 316) Equation 6.84 (p. 316) of Calculation Procedure at Pressure Inlet Boundaries (p. 315). It is recommended that pressure boundary conditions are not set to zero for compressible flows that use the ideal gas law.

  Setup → Boundary Conditions

- **Review turbulence specifications at flow boundaries. Default values detected.**

  If your case setup has any of the turbulence models enabled, be sure to review the default parameters for **K and Epsilon** or **K and Omega** for the **Turbulence Specification Method** in the outlet and inlet boundary conditions. ANSYS Fluent's default parameters for the (**Backflow**) **Turbulent Kinetic Energy**, (**Backflow**) **Turbulent Dissipation Rate**, and (**Backflow**) **Specific Dissipation Rate** are 1. You can either adjust the values, or select a different **Turbulence Specification Method**. For general information turbulence parameters, see Determining Turbulence Parameters (p. 303).

  Setup → Boundary Conditions

- **Assign non-zero layer thicknesses for wall boundaries with shell conduction.**

  When the **Shell Conduction** option is enabled in the **Wall** boundary condition dialog box, ANSYS Fluent will compute heat conduction for the wall not only in the normal direction, but also in the planar directions. To enable such computations, you must specify a nonzero **Thickness** for each layer in the **Shell Conduction Layers** dialog box. See Shell Conduction (p. 376) for information on shell conduction in thin walls.

  Setup → Boundary Conditions

- **Assign a value of 0 or 1 for VOF at the inlet or outlet boundary conditions.**

  When enabling the VOF model, the **Volume Fraction** in the inlet and outlet boundary conditions for each phase should be set either to 0 or 1. No intermediate values are permitted. For general information on boundary condition setup, see Defining Multiphase Cell Zone and Boundary Conditions (p. 1391).

  Setup → Boundary Conditions

- **Change the outlet boundary condition. Outflow boundary condition is not compatible with current multiphase settings.**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You cannot assign an outflow boundary condition when using the mixture and Eulerian multiphase models. Note the limitations of this boundary condition in Outflow Boundary Conditions (p. 350). ANSYS Fluent can model the effects of open channel flow using the VOF formulation. In such a case, outflow boundary conditions can be used at the outlet of open channel flows, to model flow exits where the details of the flow velocity and pressure are not known prior to solving the flow problem. See Open Channel Flow in the Theory Guide, under the heading Outflow Boundary, for more information.

Setup → Boundary Conditions

- **Review wall motion. Stationary wall motion relative to adjacent cell zone detected.**

  In cases where the fluid zone motion type is specified as **Moving Mesh** or **Moving Reference Frame**, all wall zones should be set to **Moving Wall** in the **Momentum** tab in the **Wall** boundary conditions dialog box. The wall motion should be defined **Relative to Adjacent Cell Zone**. The exception to this is if the walls are stationary in the absolute frame. To define wall motion, see Inputs at Wall Boundaries (p. 358).

Setup → Boundary Conditions

- **Assign non-zero velocities when specifying a moving fluid zone.**

  If selecting either **Moving Mesh** or **Moving Reference Frame** in the **Fluid** dialog box, be sure to set nonzero values for the rotational and translational velocities. Refer to Defining Zone Motion (p. 256) for user inputs.

Setup → Cell Zone Conditions

- **Review flow specifications at inlet boundaries. Default values detected.**

  For **mass-flow-inlet** and **velocity-inlet** boundary conditions, the default values in ANSYS Fluent are $1kg/s$ and $0m/s$, respectively. Review the settings and adjust accordingly. See Default Settings at Velocity Inlet Boundaries (p. 321) and Default Settings at Mass-Flow Inlet Boundaries (p. 329) for default parameters of velocity inlets and mass flow inlets, respectively.

Setup → Boundary Conditions

- **Define the porous zone when using the heat exchanger model.**

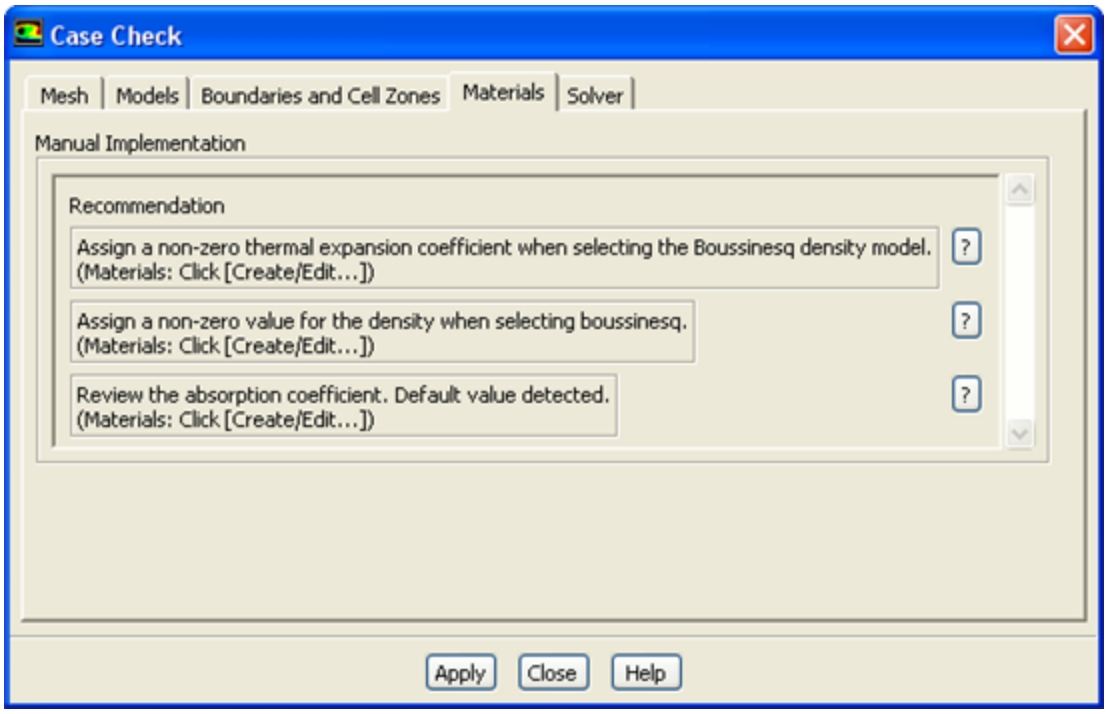
  Heat exchanger models always require the definition of the porous media zone on the primary side for the macro model and for both primary and auxiliary sides for the dual cell model. See Streamwise Pressure Drop in the Theory Guide for more information.

Setup → Cell Zone Conditions

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.20.2.4. Checking Material Properties

Figure 28.50: The Materials Tab in the Case Check Dialog Box



The following recommendations appear under the **Materials** tab (Figure 28.50: The Materials Tab in the Case Check Dialog Box (p. 1664)):

- **Assign individual fluid Cps to polynomial functions of temperature.**

  For cases with species transport and volumetric reactions, it is best practice to specify the specific heat capacity C_p as a polynomial that is a function of temperature. See Defining Properties for the Mixture and Its Constituent Species (p. 972) and Specific Heat Capacity as a Function of Temperature (p. 514) for information on defining material properties for the species in the mixture.

  Setup → **Materials**

- **Assign a non-zero value for the density when selecting boussinesq.**

  The Boussinesq model is used for natural convection problems involving small changes in temperature. To enable the Boussinesq approximation for density, choose **boussinesq** from the **Density** drop-down list in the **Create/Edit Materials** dialog box and specify a constant value for **Density**. See Inputs for the Boussinesq Approximation (p. 480).

  Setup → **Materials**

- **Review the absorption coefficient. Default value detected.**

  If any of the radiation models are enabled. Enter an absorption coefficient for the material listed (Radiation Properties (p. 515)).

  Setup → **Materials**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

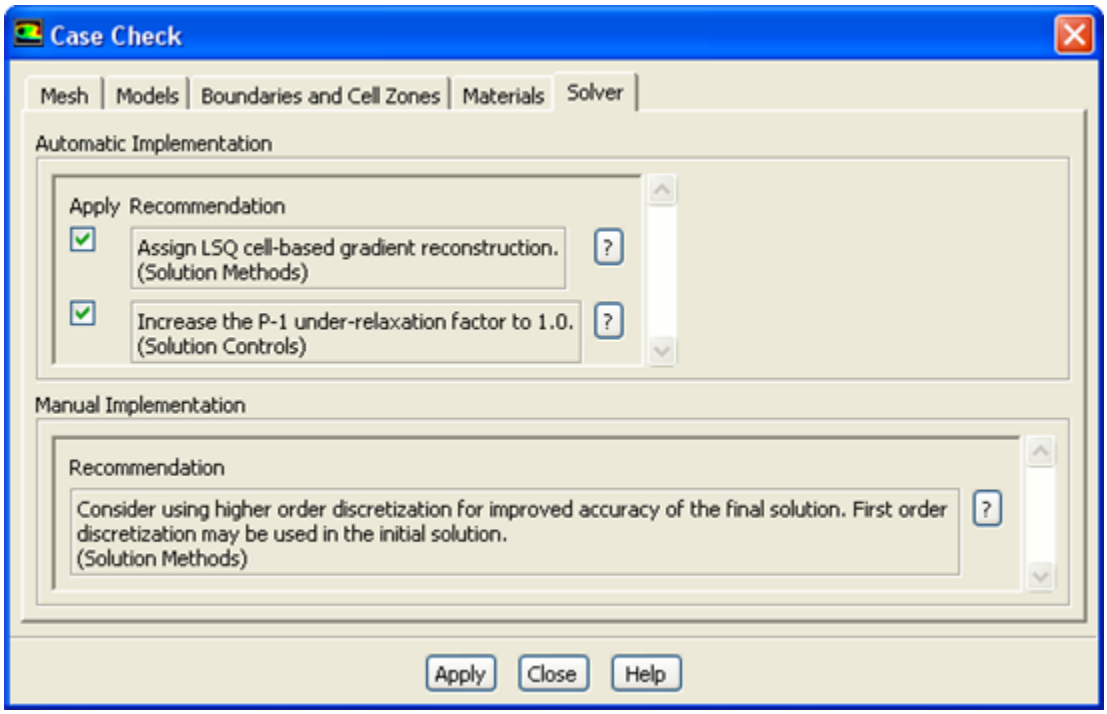
- **Assign a non-zero thermal expansion coefficient when selecting the Boussinesq density model.**

  When selecting **boussinesq** to describe the density of your material, be sure to enter a valid thermal expansion coefficient for your material. For detailed information on the Boussinesq model, see The Boussinesq Model (p. 833).

  Setup → Materials

28.20.2.5. Checking the Solver Settings

Figure 28.51: The Solver Tab in the Case Check Dialog Box



The following recommendations appear under the **Solver** tab (Figure 28.51: The Solver Tab in the Case Check Dialog Box (p. 1665)):

- **Enable the unsteady solver option when selecting moving mesh for the fluid boundary.**

  If the motion type of the fluid boundary condition is specified as **Moving Mesh**, then your case should be specified as **Transient** in the **General** task page. Visit Setting Up the Sliding Mesh Problem (p. 634) for steps on setting up moving mesh problem.

  Setup → General

- **Assign LSQ cell-based gradient reconstruction.**

  The least squares cell-based averaging scheme is known to be as accurate as the node-based gradient for irregular unstructured meshes, but less expensive to compute than the node-based gradient. Therefore, it is recommended that least squares cell-based gradient reconstruction is used. See Evaluation of Gradients and Derivatives in the Theory Guide for more information on gradient options.

  Solution → Methods

- **Change the under-relaxation factor for the energy equation to at least 0.90.**

  You should set the energy under-relaxation factor between 0.90 and 1.0. If you decide to apply this recommendation, then ANSYS Fluent will automatically set the energy under-relaxation factor to `0.90`. If you want to increase this value, you can manually make the change by going to the **Solution Controls** task page. See Solution Strategies for Heat Transfer Modeling (p. 828) for the under-relaxation of the energy equation.

  Solution → **Controls**

- **Increase the NOx under-relaxation factor to at least 0.90.**

  If the NOx model is enabled, set the NOx under-relaxation factor to a value of at least 0.90 to fully converge the solution. Note that the under-relaxation factor could be lower at the start of the run, but can then be increased after an initial solution is obtained. If you decide to apply this recommendation, then ANSYS Fluent will automatically set the NOx under-relaxation factor to `0.90`. If you want to increase this value, you can manually make the change by going to the **Solution Controls** task page. See Using the NOx Model (p. 1153).

  Solution → **Controls**

- **Increase the Discrete Ordinates under-relaxation factor to at least 0.90.**

  If the Discrete Ordinates (DO) radiation model is enabled, set the radiation under-relaxation factor to a value of at least 0.90 to fully converge the solution. Note that the under-relaxation factor could be lower at the start of the run, but can then be increased after an initial solution is obtained. If you decide to apply this recommendation, then ANSYS Fluent will automatically set the radiation under-relaxation factor to `0.90`. If you want to increase this value, you can manually make the change by going to the **Solution Controls** task page. See DO Solution Parameters (p. 880).

  Solution → **Controls**

- **Increase the P1 under-relaxation factor to 1.0.**

  If the P1 radiation model is enabled, set the radiation under-relaxation factor to 1.0 to fully converge the solution. Note that the under-relaxation factor could be lower at the start of the run, but can then be increased after an initial solution is obtained. If you decide to apply this recommendation, then ANSYS Fluent will automatically set the radiation under-relaxation factor to `1.0`. See P-1 Model Solution Parameters (p. 878).

  Solution → **Controls**

- **Increase the species and energy under-relaxation factors to at least 0.90.**

  For a case with species transport and energy defined, set the species and energy under-relaxation factors to a value of at least 0.90. If you decide to apply this recommendation, then ANSYS Fluent will automatically set the species and energy under-relaxation factors to `0.90`. If you want to increase this value, you can manually make the change by going to the **Solution Controls** task page. See Solution Procedures for Chemical Mixing and Finite-Rate Chemistry (p. 990).

  Solution → **Controls**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Assign a value of 1 for the under-relaxation factor for unsteady DPM with 1 DPM update per time step.**

  It is recommended that the DPM under-relaxation factor be set to 1 for unsteady DPM with 1 DPM update per time step.

  Solution → **Controls**

- **Increase the mean mixture fraction under-relaxation factor to at least 0.90.**

  If the non-premixed or partially premixed combustion models are enabled, then it is best to set the mean mixture under-relaxation factor to a value of at least 0.90 to ensure full convergence. If you decide to apply this recommendation, then ANSYS Fluent will automatically set the mean mixture under-relaxation factor to `0.90`. If you want to increase this value, you can manually make the change by going to the **Solution Controls** task page. See Solving the Flow Problem (p. 1083).

  Solution → **Controls**

- **Consider using higher order discretization for improved accuracy of the final solution. First-order discretization may be used in the initial solution.**

  It is generally advisable to obtain an initial solution using first-order accurate discretization, however, second order discretization is recommended for improved accuracy of the final solution. See Choosing the Spatial Discretization Scheme (p. 1554) for more information on discretization schemes.

  Solution → **Methods**

- **Select the absolute reference frame for initializing cases when using the MRF model.**

  When using the MRF model, always use the absolute reference frame while initializing the solution. Select **Absolute** under **Reference Frame** in the **Solution Initialization** task page. If the **Relative to Cell Zone** option is selected, which is the default option, the initial flow field can contain discontinuities, which can cause convergence problems in the first few iterations. Refer to Initializing the Entire Flow Field Using Standard Initialization (p. 1591) for more information.

  Solution → **Initialization** **Initialize**

- **Choose PRESTO! for the pressure discretization scheme.**

  When using the VOF model, it is recommended that you use **PRESTO!** as the pressure discretization scheme. This scheme is recommended for flows with high swirl numbers, a high-Rayleigh-number natural convection, high-speed rotating flows, flows involving porous media, and flows in strongly curved domains. See Choosing the Pressure Interpolation Scheme (p. 1556) for more information.

  Solution → **Methods**

28.21. Convergence and Stability

Convergence can be hindered by a number of factors. Large numbers of computational cells, overly conservative under-relaxation factors, and complex flow physics are often the main causes. Sometimes it is difficult to know whether you have a converged solution. In the following sections, some of the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

numerical controls and modeling techniques that can be exercised to enhance convergence and maintain stability are examined.

28.21.1. Judging Convergence
28.21.2. Step-by-Step Solution Processes
28.21.3. Modifying Algebraic Multigrid Parameters
28.21.4. Modifying the Multi-Stage Parameters
28.21.5. Robustness on Meshes of Poor Quality
28.21.6. Warped-Face Gradient Correction

You should also refer to Choosing the Spatial Discretization Scheme (p. 1554) and Choosing the Pressure-Velocity Coupling Method (p. 1561) for information about how the choice of discretization scheme or (for the pressure-based solver) pressure-velocity coupling scheme can affect convergence. Manipulation of under-relaxation parameters and multigrid settings to enhance convergence is discussed in Setting Under-Relaxation Factors (p. 1564) and Modifying Algebraic Multigrid Parameters (p. 1670).

28.21.1. Judging Convergence

There are no universal metrics for judging convergence. Residual definitions that are useful for one class of problem are sometimes misleading for other classes of problems. Therefore it is a good idea to judge convergence not only by examining residual levels, but also by monitoring relevant integrated quantities such as drag or heat transfer coefficient.

For most problems, the default convergence criterion in ANSYS Fluent is sufficient. This criterion requires that the globally scaled residuals, defined by Equation 28.6 (p. 1625) or Equation 28.12 (p. 1626) decrease to 10^{-3} for all equations except the energy and P-1 equations, for which the criterion is 10^{-6}. Locally scaled residuals defined by Equation 28.7 (p. 1625) or Equation 28.13 (p. 1626) decrease to 10^{-5} for all equations.

Sometimes, however, this criterion may not be appropriate. Typical situations are listed below.

- If you make a good initial guess of the flow field, the initial continuity residual may be very small leading to a large scaled residual for the continuity equation. In such a situation it is useful to examine the unscaled residual and compare it with an appropriate scale, such as the mass flow rate at the inlet.

- For some equations, such as for turbulence quantities, a poor initial guess may result in high scale factors. In such cases, scaled residuals will start low, increase as nonlinear sources build up, and eventually decrease. It is therefore good practice to judge convergence not just from the value of the residual itself, but from its behavior. You should ensure that the residual continues to decrease (or remain low) for several iterations (say 50 or more) before concluding that the solution has converged.

Another popular approach to judging convergence is to require that the unscaled residuals drop by three orders of magnitude. ANSYS Fluent provides residual normalization for this purpose, as discussed in Definition of Residuals for the Pressure-Based Solver (p. 1624), where residuals are defined for both the pressure-based solver and the density-based solver. In this approach the convergence criterion is that the normalized unscaled residuals should drop to 10^{-3}. However, this requirement may not be appropriate in many cases:

- If you have provided a very good initial guess, the residuals may not drop three orders of magnitude. In a nearly-isothermal flow, for example, energy residuals may not drop three orders if the initial guess of temperature is very close to the final solution.

- If the governing equation contains nonlinear source terms which are zero at the beginning of the calculation and build up slowly during computation, the residuals may not drop three orders of magnitude. In the case

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of natural convection in an enclosure, for example, initial momentum residuals may be very close to zero because the initial uniform temperature guess does not generate buoyancy. In such a case, the initial nearly-zero residual is not a good scale for the residual.

- If the variable of interest is nearly zero everywhere, the residuals may not drop three orders of magnitude. In fully-developed flow in a pipe, for example, the cross-sectional velocities are zero. If these velocities have been initialized to zero, initial (and final) residuals are both close to zero, and a three-order drop cannot be expected.

In such cases, it is wise to monitor integrated quantities, such as drag or overall heat transfer coefficient, before concluding that the solution has converged. It may also be useful to examine the un-normalized unscaled residual, and determine if the residual is small compared to some appropriate scale. Alternatively, the scaled residual defined by Equation 28.6 (p. 1625) or Equation 28.12 (p. 1626) (the default) may be considered.

Conversely, it is possible that if the initial guess is very bad, the initial residuals are so large that a three-order drop in residual does not guarantee convergence. This is specially true for k and ε equations where good initial guesses are difficult. Here again it is useful to examine overall integrated quantities that you are particularly interested in. If the solution is unconverged, you may drop the convergence tolerance, as described in Modifying Convergence Criteria (p. 1631).

28.21.2. Step-by-Step Solution Processes

One important technique for speeding convergence for complex problems is to tackle the problem one step at a time. When modeling a problem with heat transfer, you can begin with the calculation of the isothermal flow. To solve turbulent flow, you might start with the calculation of laminar flow. When modeling a reacting flow, you can begin by computing a partially converged solution to the non-reacting flow, possibly including the species mixing. When modeling a discrete phase, such as fuel evaporating from droplets, it is a good idea to solve the gas-phase flow field first. Such solutions generally serve as a good starting point for the calculation of the more complex problems. These step-by-step techniques involve using the Solution Controls Task Page (p. 2508) to turn equations on and off in the **Equations** dialog box.

28.21.2.1. Selecting a Subset of the Solution Equations

ANSYS Fluent automatically solves each equation that is turned on using the **Models** family of dialog boxes. If you specify in the Viscous Model Dialog Box (p. 2176) that the flow is turbulent, equations for conservation of turbulence quantities are turned on. If you specify in the Energy Dialog Box (p. 2175) that ANSYS Fluent should enable energy, the energy equation is enabled. Convergence can be sped up by focusing the computational effort on the equations of primary importance. The **Equations** list in the Equations Dialog Box (p. 2511) allows you to turn individual equations on or off temporarily.

Solution → **Controls** → **Equations...**

A typical example is the computation of a flow with heat transfer. Initially, you will define the full problem scope, including the thermal boundary conditions and temperature-dependent flow properties. Following the problem setup, you will use the **Equations** dialog box to temporarily turn off the energy

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

equation. You can then compute an isothermal flow field, remembering to set a reasonable initial value for the temperature of the fluid.

Important

This is possible only for the pressure-based solver; the density-based solver solves the energy equation together with the flow equations in a coupled manner, so you cannot turn off the energy equation as described above.

When the isothermal flow is reasonably well converged, you can turn the energy equation back on. You can actually turn off the momentum and continuity equations while the initial energy field is being computed. When the energy field begins to converge well, you can turn the momentum and continuity equations back on so that the flow pattern can adjust to the new temperature field. The temperature will couple back into the flow solution by its impact on fluid properties such as density and viscosity. The temperature field will have no effect on the flow field if the fluid properties (for example, density, viscosity) do not vary with temperature. In such cases, you can compute the energy field without turning the flow equations back on again.

Important

If you have specified temperature-dependent flow properties, you should be sure that a realistic value has been set for temperature throughout the domain before disabling calculation of the energy equation. If an unrealistic temperature value is used, the flow properties dependent on temperature will also be unrealistic, and the flow field will be adversely affected. Instructions for initializing the temperature field or patching a temperature field onto an existing solution are provided in Initializing the Solution (p. 1591).

28.21.2.2. Turning Reactions On and Off

To solve a species mixing problem prior to solving a reacting flow, you should set up the problem including all of the reaction information, and save the complete case file. To turn off the reaction so that only the species mixing problem can be solved, you can use the Species Model Dialog Box (p. 2216) to turn off the **Volumetric** option under **Reactions**.

Setup → **Models** → **Species** → **Edit...**

Once the species mixing problem has partially converged, you can return to the **Species Model** dialog box and turn the **Volumetric Reactions** option on again. You can then resume the calculation starting from the partially converged data.

For combustion problems you may want to patch a hot temperature in the vicinity of the anticipated reactions before you restart the calculation. See Patching Values in Selected Cells (p. 1593) for information about patching an initial value for a flow variable.

28.21.3. Modifying Algebraic Multigrid Parameters

The default algebraic multigrid settings are appropriate for nearly all problems, but in rare cases you may need to make minor adjustments. Setting Algebraic Multigrid Parameters (p. 1577) describes how to analyze the multigrid solver's performance to determine which parameters.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.21.4. Modifying the Multi-Stage Parameters

It is possible to make several changes to the multi-stage time-stepping scheme itself. See Changing the Multi-Stage Scheme (p. 1588) for detailed information.

28.21.5. Robustness on Meshes of Poor Quality

Poor quality meshes are meshes containing highly skewed cells, highly non-orthogonal cells, non-convex cells, or cells with left-handed faces. Such mesh elements tend to decrease the numerical stability of traditional CFD discretization algorithms. These mesh elements require special treatment, namely, a numerical correction of the transport equation discretization, which is intended to improve the numerical properties of the solution algorithms at mesh cells of poor quality.

Note

If your mesh exhibits any type of poor quality except left-handed faces, you can maintain second order accuracy by using the warped-face gradient correction and disabling poor mesh numerics. To disable poor mesh numerics, set `solve/set/poor-mesh-numerics/enable?` to `no`. For additional information on this method, see Warped-Face Gradient Correction (p. 1674).

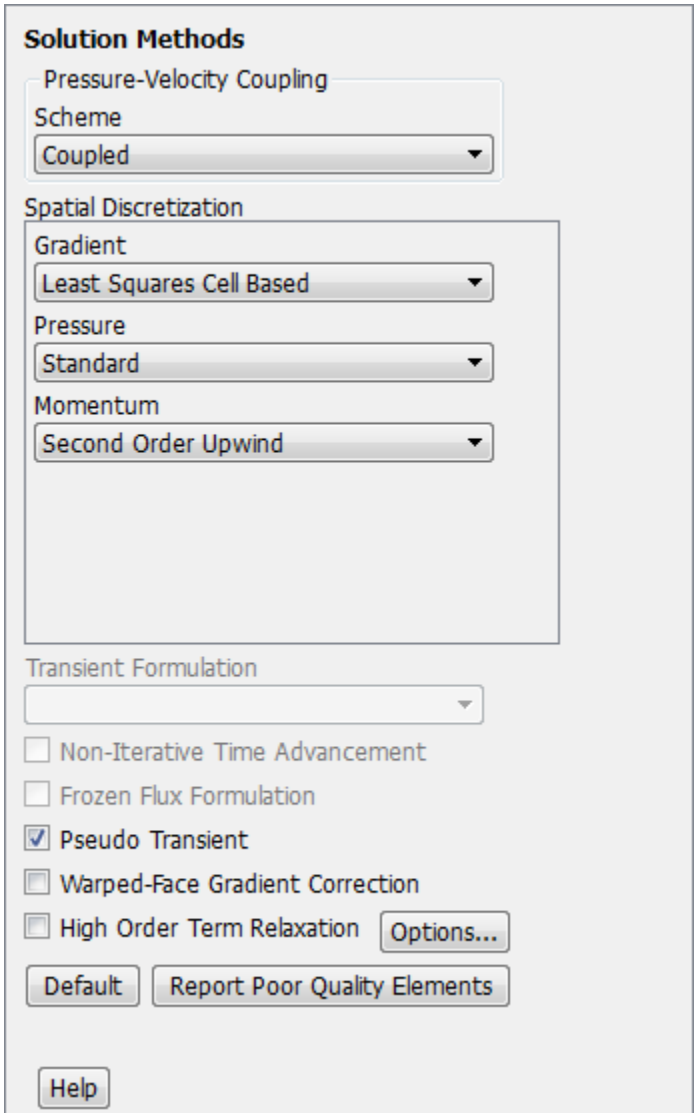
In order to facilitate solution convergence on meshes of poor quality, the ANSYS Fluent solver can apply a local solution correction, limited spatially to distorted cells of the mesh. If you read in a poor quality mesh containing cells and faces with corrupt metrics, you will see a warning in the TUI of the form:

```
Info: The mesh contains elements that are invalid or of poor quality.

      A different numerical scheme will be applied to these elements,
      which may affect the quality of the solution. It is recommended
      that you consider removing the invalid and poor quality elements
      in the mesh.

      For more information on the invalid and poor quality elements,
      please use the following TUI commands:

      /mesh/check
      /mesh/repair-improve/report-poor-elements.
```

Additionally, in the **Solution Methods** task page, the **Report Poor Quality Elements** button will appear, which can be used to report more statistics on the number and type of cells that ANSYS Fluent has identified as having poor quality (see Figure 28.52: Reporting Poor Quality Elements (p. 1672)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.52: Reporting Poor Quality Elements



In order to facilitate solution convergence on meshes of poor quality, the ANSYS Fluent solver can apply a local solution correction, limited spatially to distorted cells of the mesh. The corrected solution can be of 0th, 1st, or 2nd order:

- The 0th order scheme applies an algorithm that computes the solution variable for the transport equation in the bad cells by assembling the solution directly from the surrounding solution in the better quality cells.
- The 1st order scheme applies locally low order discretization methods and neglects some non-orthogonal contributions to the gradients when computing the diffusive fluxes.
- The 2nd order scheme only modifies the numerics in the bad cells by assembling the gradient vector for the given solution variable from the gradients in the surrounding better quality cells.

In other words, the discretization error will be independent of the mesh size in this region if 0th order is used. It will decrease linearly with subsequent grid refinement when 1st order is used, and quadratically when the 2nd order option is selected.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

By default, Fluent will apply the 1st order correction scheme to any invalid cells. Using text commands, you can further control the application of the poor mesh numerics by using one of the other schemes, including highly skewed and highly non-orthogonal cells, and including cells from a user-defined register. The text commands that are available are as follows:

/solve/set/poor-mesh-numerics/enable?

Choose whether or not to apply the poor mesh numerics corrections and specify which order scheme to use.

In general, it is recommended that you use the 1st order solution correction, which provides a reasonable compromise between accuracy and stability gain. For meshes of better (and yet still low) quality it is advisable to try the 2nd order option, which will preserve the mesh convergence behavior provided by the convection term discretization schemes. In case no convergence is obtained with any of the aforementioned schemes, you should use the 0th order option, which will provide the highest stability and at the same time the lowest accuracy. In regions with highly nonlinear flow physics this scheme can yield highly nonphysical results.

The computational time (or cost or run time requirements) for all three schemes will increase linearly as the number of cells identified as poor quality cells increases. Provided that there is reasonable convergence behavior with all three schemes, the 1st order option will be the fastest; 0th and 2nd order schemes will have similar run times to each other.

/solve/set/poor-mesh-numerics/cell-quality-based?

Choose whether or not the poor mesh numerics should also be applied to cells that are highly non-orthogonal or highly skewed.

/solve/set/poor-mesh-numerics/set-quality-threshold

Set the threshold for including cells based on cell quality metrics when `cell-quality-based?` is enabled. The value you enter is applied as a threshold for the cell orthogonality and the complement of cell skewness below which the poor mesh numerics will be applied. The default value is `0.05`. Some cases may require tuning this parameter to improve stability.

/solve/set/poor-mesh-numerics/user-defined-on-register

Answer **yes** at the prompt to Include cells from a register in the poor mesh numerics treatment. You will be prompted for the name or ID of the register to include. Note that if you answer **no** at the prompt any cells previously included from registers are removed from the list of cells for poor mesh numerics.

If you include a user-defined register in the poor mesh numerics, reporting of poor mesh element statistics will also include the cells added by the `user-defined-on-register` command. For example:

```
Poor Mesh Element Statistics:
Identified    0 faces with too small area.
Identified    0 faces adjacent to negative volume cells.
```

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Identified    0 faces adjacent to bad quality cells.
Identified   43 cells from user-defined register.
```

Note

The statistics not only report the cells included in the register, but also their neighboring cells, resulting in an overall larger number of cells being identified.

Important

It is your responsibility to maintain the register-based cells. If you change the mesh, (for example, using mesh manipulation, adaptation, and so on), then you must use the `user-defined-on-register` command again. Enter `yes` to the command prompt in order to update the cells based on a new register, otherwise, enter `no` to remove all cells from the poor mesh numerics list. For more information about registers, see Registers (p. 1700).

`/solve/set/poor-mesh-numerics/print-poor-elements-count`

Print out a listing of the poor cells for each criterion (default, cell quality, and user-defined).

```
> /solve/set/poor-mesh-numerics/print-poor-elements-count

- Number of marked cells based on default criteria       = 5
- Number of marked cells based on cell quality criteria  = 11
- Number of marked cells based on user-defined criteria  = 0
```

`/solve/set/poor-mesh-numerics/reset-poor-elements?`

Reset the poor cells included from user-defined registers and cell-quality metrics. The poor cells identified from the internal mesh check will be retained.

28.21.6. Warped-Face Gradient Correction

The warped-face gradient correction (WFGC) is designed to improve gradient accuracy for all gradient methods—cell based, node based and least squares based gradients. WFGC is recommended for 3D simulations on polyhedral, hexahedral, and hybrid meshes (meshes containing a mix of hexahedral and polyhedral cells), and it is enabled by default for any mesh that contains polyhedra. Note that these types of mesh cells may not have perfectly planar faces, which can affect the overall accuracy of the Green Gauss surface integral.

WFGC computations aim to resolve gradient accuracy degradation due to very high aspect ratio, cells with non-flat faces in the boundary layer, and highly deformed cells where the formal cell centroid lies outside of the control volume.

WFGC has two modes, fast mode and memory saving mode. Fast mode does not increase gradient computation time for static meshes (except for cases that use the **Green-Gauss Node Based** gradient method with a turbulence model), but it does require additional memory. There is roughly a 22% increase in memory for a basic flow solution, but this additional overhead does not change as more models

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(turbulence, species transport, radiation, and so on) are enabled. Memory saving mode does not increase memory consumption, but it slows down iteration execution time for static and dynamic meshes.

Important

Enabling **Warped-Face Gradient Correction** through the **Solution Methods** task page enables fast mode. To switch to memory saving mode, you must use the TUI command `solve/set/warped-face-gradient-correction/enable?` and enter `no` in response to the `use fast mode?` prompt.

WFGC does *not* correct for cells with left-handed faces. If your mesh contains left-handed faces, you must use poor mesh numerics to obtain a stable solution.

28.22. Solution Steering

For additional information, see the following sections:

28.22.1. Overview of Solution Steering
28.22.2. Solution Steering Strategy
28.22.3. Using Solution Steering

28.22.1. Overview of Solution Steering

Solution steering in the density-based implicit solver provides you with an expert system that will help navigate the flow solution from a starting initial guess to a converged solution with minimum user interaction. When you apply solution steering, you will be required to select the type of flow that best characterizes the solution domain and the maximum desired accuracy, and then allow the solver to take the solution to convergence. As the solver proceeds with the solution iteration, certain solver parameters will be adjusted behind the scenes to insure that a converged solution to steady-state is possible.

Important

Solution steering is available only for steady-state flows in the density-based implicit solver.

28.22.2. Solution Steering Strategy

The convergence to steady-state solution is achieved in two stages. The parameters that are used in these stages are determined and set based on user input for the type of flow that can best characterize the solution domain. The type of flows available for selection are classified based on flow compressibility as well as the dominant flow Mach number in the solution domain.

The following flow types are available:

- Incompressible (if the flow is incompressible, that is density is constant)
- Subsonic (if the flow is compressible and M<0.75)
- Transonic (if the flow is compressible and 0.65<M<1.2)
- Supersonic (if the flow is compressible and 1.10< M<2.5)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Hypersonic (if the flow is compressible and 2.0< M)

Important

There is no exact Mach number cut-off for these regions, therefore, the above Mach number ranges are just a simple guideline to help you select a flow type.

Solution steering will typically perform full multigrid (FMG) initialization followed by two iterative stages. The purpose of each stage is described below.

28.22.2.1. Initialization

Immediately before the start of the iteration, solution steering will perform full multigrid initialization to obtain the best possible initial starting solution.

Stage 1:

The purpose of Stage 1 is to navigate the solution from the difficult initial phase of the solution toward convergence by insuring maximum stability. During this stage, the solution is advanced gradually from 1st-order accuracy to maximum accuracy (user specified and typically 2nd-order) at a constant low CFL value.

Stage 2:

In this stage the solution is driven hard towards convergence by regular adjustments of the CFL value to insure fast convergence as well as to prevent possible divergence.

In stage 2, the residual history is monitored and analyzed through regular intervals to determine if an increase or decrease in CFL value is needed to obtain fast convergence or to prevent divergence.

28.22.3. Using Solution Steering

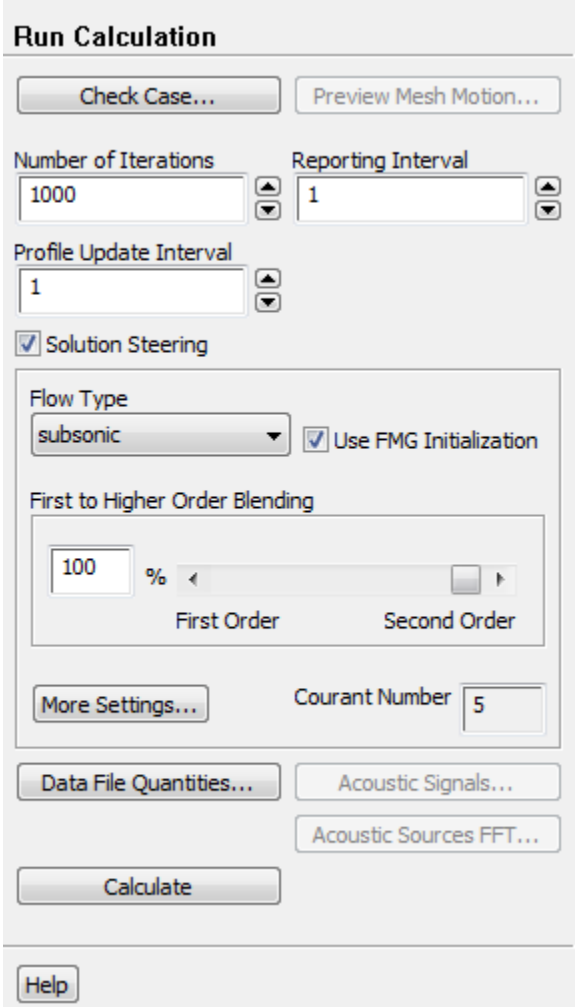
Solution steering is disabled by default. However, when the following criteria are met, the solution steering feature will become available for selection:

Feature	Setting
Solver Type	Density Based
Solver Formulation	Implicit
Time Formulation	Steady
Data is valid (either data file has been read or flow has been initialized)	

To turn on solution steering, enable the **Solution Steering** option as shown in Figure 28.53: The Run Calculation Task Page with Solution Steering Enabled (p. 1677).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.53: The Run Calculation Task Page with Solution Steering Enabled



The **Run Calculation** task page will then expand to display the solution steering main controls (see Figure 28.53: The Run Calculation Task Page with Solution Steering Enabled (p. 1677)). To obtain a flow solution using solution steering, you will need to perform the following:

1. Select the type of flow.
2. Select the maximum accuracy desired (first to second order blending).
3. Click **Calculate**.

You can also adjust the number of iterations, or customize the parameters of the solution steering if the default setting is not sufficient for the type of flow problem being solved.

Before using solution steering, you will need to prepare and set up the case as usual as described in the Getting Started Manual.

After **Solution Steering** is enabled, specify the following:

Flow Type

allows you to select the flow type that best describes the flow in the solution domain. Five choices are available: **incompressible**, **subsonic**, **transonic**, **supersonic**, and **hypersonic**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

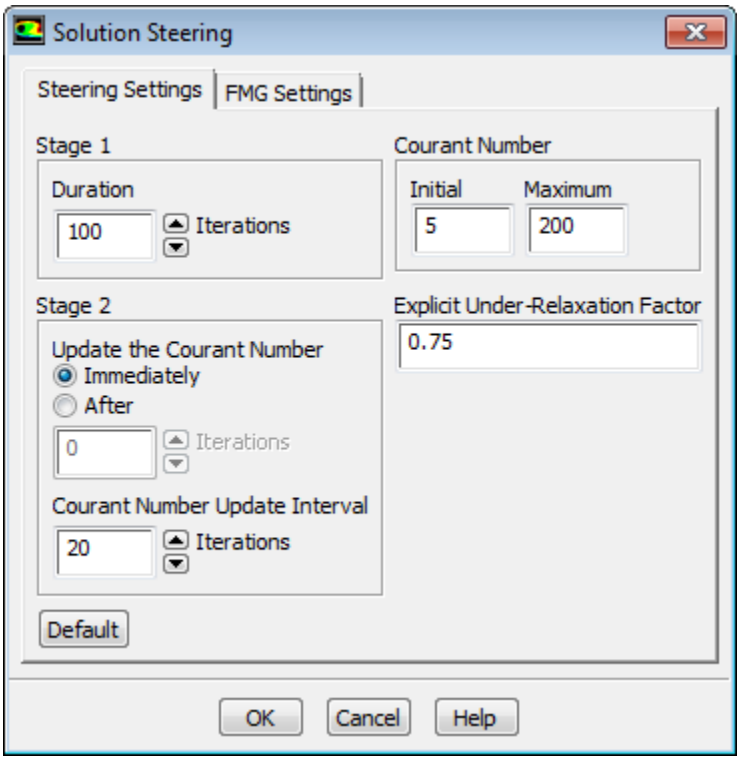
FMG Initialization
when enabled allows for full multigrid initialization before starting stages 1 and 2. FMG initialization is enabled by default.

First to Higher Order Blending
allows you to reduce the desired solution accuracy by selecting a blending factor less than 100%. The default setting is 100%. See First-to-Higher Order Blending in the Theory Guide for more information. The blending factor will be grayed out if **Second Order Upwind** discretization for the **Flow** equations is not selected in the **Solution Methods** task page. The solution accuracy may be reduced (typical values are 75% or 50%) if it is not possible to obtain a converged solution with the maximum second-order accuracy (that is blending = 100%)

Courant Number
in the **Run Calculation** task page is a non-adjustable field displaying the current CFL number, which allows you to view it during the calculation.

More Settings...
opens the **Solution Steering** dialog box, providing a host of settings that control the solution steering strategy, as shown in Figure 28.54: The Solution Steering Dialog Box (p. 1678).

Figure 28.54: The Solution Steering Dialog Box



The **Solution Steering** dialog box, shown in Figure 28.54: The Solution Steering Dialog Box (p. 1678), contains two tabs. The **Steering Settings** tab sets the solution steering parameters and the **FMG Settings** sets the full multigrid initialization parameters.

In the **Steering Setting** tab, you can modify the parameters used in Stages 1 and 2.

Stage 1
Duration is the number of iterations in stage 1. The CFL number used during these iterations is set in the **Initial** field, in the **Courant Number** group box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stage 2

The Courant number update in stage 2 can start immediately after the end of stage 1, or after a certain designated number of iterations. If the Courant number update is to start immediately after stage 1 then **Immediately** should be selected (this is the default option). If the Courant number update is desired after some lagged period of iterations, then **After** should be selected and the lag in the number of iterations should be entered in the field below it. The frequency at which the Courant number is updated is defined in **Courant Number Update Interval** field.

Courant Number

Initial is the starting Courant number and **Maximum** is the maximum allowed Courant number. The solution steering algorithm will not allow the solver to exceed the maximum Courant number, but will allow the solver to use a Courant number less than the initial Courant number if divergence in the solution has occurred.

Explicit Under-Relaxation Factor

allows the solution to be under-relaxed to improve convergence. The under-relaxation value is determined by the **Flow Type** that you selected in the **Run Calculation** task page, when **Solution Steering** was enabled. In general, you do not need to alter the default value set in this field. Refer to Under-Relaxation of Variables in the Theory Guide for more information about explicit relaxation.

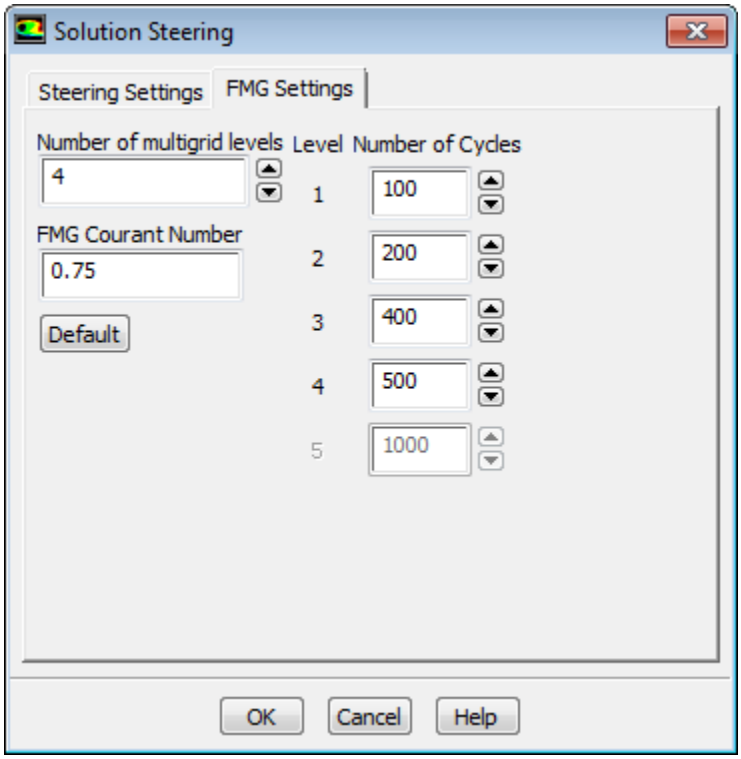
Default

is available in the **Steering Settings** tab to reset any changes made to the parameters to their original default values.

In the **FMG Settings** tab (Figure 28.55: The FMG Settings Tab in the Solution Steering Dialog Box (p. 1680)), the **Number of multigrid levels** and **Number of Cycles** in each **Level**, as well as the **FMG Courant Number** used in the FMG initialization can be adjusted. The default values used in the multigrid settings are determined from the type of flow that you selected, the size of the mesh, and the flow dimensionality. The **Default** button is used to reset any changes to the original default values. For more information about FMG initialization, refer to Full Multigrid (FMG) Initialization (p. 1595).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.55: The FMG Settings Tab in the Solution Steering Dialog Box



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 29: Adapting the Mesh

The solution-adaptive mesh refinement feature of ANSYS Fluent allows you to refine and/or coarsen your mesh based on geometric and numerical solution data. In addition, ANSYS Fluent provides tools for creating and viewing adaption fields customized to particular applications. For information about the theory behind mesh adaption in ANSYS Fluent, see Adapting the Mesh in the Theory Guide. Information about using the adaption process in ANSYS Fluent is described in detail in the following sections.

29.1. Using Adaption
29.2. Boundary Adaption
29.3. Gradient Adaption
29.4. Dynamic Gradient Adaption
29.5. Isovalue Adaption
29.6. Region Adaption
29.7. Volume Adaption
29.8. Yplus/Ystar Adaption
29.9. Anisotropic Adaption
29.10. Geometry-Based Adaption
29.11. Registers
29.12. Mesh Adaption Controls

29.1. Using Adaption

Two significant advantages of the unstructured mesh capability in ANSYS Fluent are:

- reduced setup time compared to structured meshes
- the ability to incorporate solution-adaptive refinement of the mesh

By using solution-adaptive refinement, you can add cells where they are needed in the mesh, thus enabling the features of the flow field to be better resolved. When adaption is used properly, the resulting mesh is optimal for the flow solution because the solution is used to determine where more cells need to be added. Thus, computational resources are not wasted by the inclusion of unnecessary cells, as occurs in the structured mesh approach. Also, the effect of mesh refinement on the solution can be studied without completely regenerating the mesh.

Note

When you perform mesh adaption in a parallel computation, a load balancing step will be performed by ANSYS Fluent by default.

The automatic load balancing will not occur in conjunction with dynamic adaption. See Dynamic Gradient Adaption (p. 1690) for information on dynamic adaption, and Load Balancing (p. 2044) for information on load balancing in parallel ANSYS Fluent. For information about the static adaption process, see Static Adaption Process in the Theory Guide.

For additional information, see the following sections:

29.1.1. Adaption Example

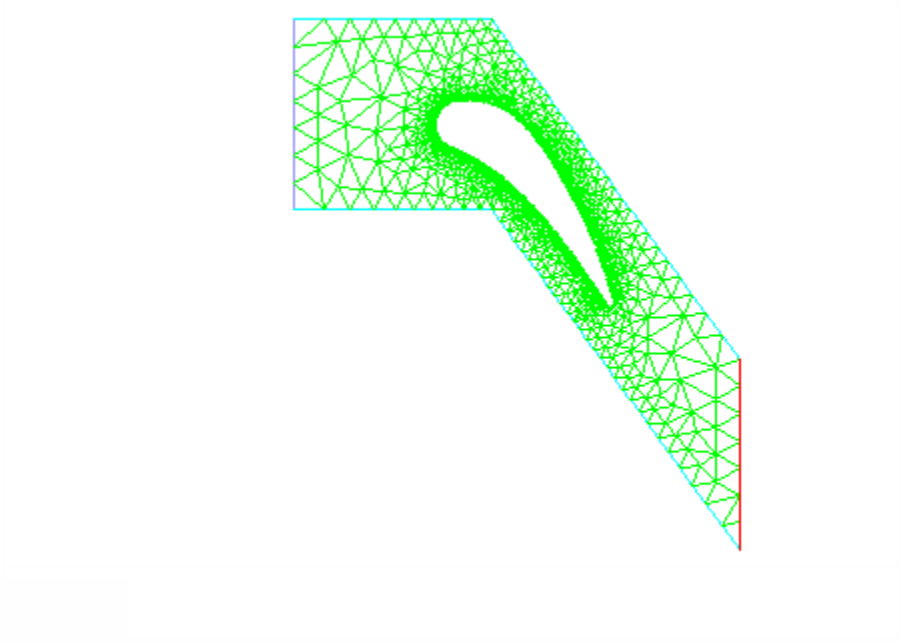
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

29.1.2. Adaption Guidelines

29.1.1. Adaption Example

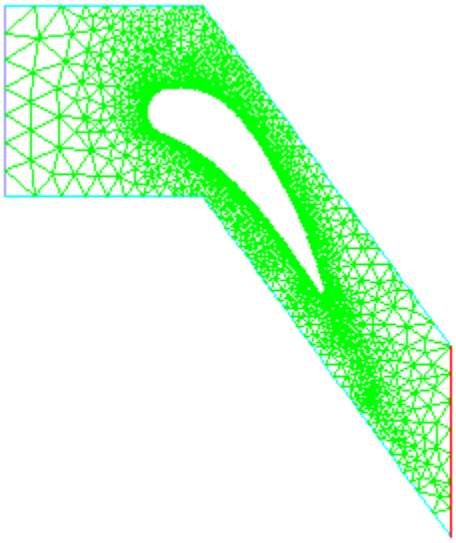
An example of the effective use of adaption is in the solution of the compressible, turbulent flow through a 2D turbine cascade. The initial mesh around the blade is fine, as shown in Figure 29.1: Turbine Cascade Mesh Before Adaption (p. 1682). The surface node distribution thus provides adequate definition of the blade geometry, and enables the turbulent boundary layer to be properly resolved without further adaption. On the other hand, the mesh on the inlet, outlet, and periodic boundaries is comparatively coarse. To ensure that the flow in the blade passage is appropriately resolved, solution-adaptive refinement was used to create the mesh shown in Figure 29.2: Turbine Cascade Mesh after Adaption (p. 1683).

Figure 29.1: Turbine Cascade Mesh Before Adaption



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 29.2: Turbine Cascade Mesh after Adaption



While the procedure for solution adaption will vary according to the flow being solved, it is instructive to examine the adaption process used for the turbine cascade shown in the previous figure. Though this example involves compressible flow, the general procedure is applicable for incompressible flows as well.

1. Display contours of pressure adaption function to determine a suitable refinement threshold (see Gradient Adaption (p. 1688)).

2. "Mark the cells within the refinement threshold, creating a refinement register (see Adaption Registers in the Theory Guide and Gradient Adaption (p. 1688)).

3. Repeat the process described in steps 1 and 2, using gradients of Mach number as a refinement criterion.

4. To refine in the wake region, use isovalues of total pressure as a criterion (see Isovalue Adaption (p. 1691)). This causes cells within the boundary layer and the wake to be marked, since these are both regions of high total-pressure loss.

5. Use the Manage Adaption Registers Dialog Box (p. 2728) to combine the three refinement registers into a single register (see Manipulating Adaption Registers (p. 1701)).

6. Limit the minimum cell volume for adaption to prevent the addition of cells within the boundary layer, where the mesh was judged to be fine enough already (see Mesh Adaption Controls Dialog Box (p. 2730)).

7. Refine the cells contained in the resulting adaption register (see Manipulating Adaption Registers (p. 1701)).

8. Perform smoothing using the Improve Mesh Dialog Box (p. 2715); see Improving the Mesh by Smoothing and Swapping (p. 230) for details.

The effect of refining on gradients is evident in the finer mesh ahead of the leading edge of the blade and within the blade passage (Figure 29.2: Turbine Cascade Mesh after Adaption (p. 1683)). The finer mesh in the wake region is due to the adaption using isovalues of total pressure.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

29.1.2. Adaption Guidelines

The advantages of solution-adaptive refinement, when used properly (as in the turbine cascade example in Adaption Example (p. 1682)), are significant. However, this capability must be used carefully to avoid certain pitfalls. Some guidelines for proper usage of solution-adaptive refinement are as follows:

- The surface mesh must be fine enough to adequately represent the important features of the geometry.

  For example, it would be bad practice to place too few nodes on the surface of a highly-curved airfoil, and then use solution refinement to add nodes on the surface. The surface will always contain the facets contained in the initial mesh, regardless of the additional nodes introduced by refinement.

- The initial mesh should contain sufficient cells to capture the essential features of the flow field.

  Consider the following example, in which you want to predict the shock forming around a bluff body in supersonic flow. To obtain a reasonable first solution, the initial mesh should contain enough cells and also have sufficient resolution to represent the shape of the body. Subsequent gradient adaption can be used to sharpen the shock and to establish a mesh-independent solution.

- Polyhedral cells are not eligible for adaption. The presence of polyhedral cells in a mesh may or may not limit the eligibility of other cells for adaption, depending on the manner in which the polyhedral cells were created:

  – If the domain was converted to polyhedra (see Converting the Domain to a Polyhedra (p. 199)), then no part of the mesh can be adapted (even if hexahedral cells are present in the mesh after conversion).

  – If the polyhedra are a result of converting skewed tetrahedral cells (see Converting Skewed Cells to Polyhedra (p. 204)) or converting the transitional cells of a hexcore mesh, then the nonpolyhedral cells may be adapted. The polyhedral cells, however, will be automatically unmarked from the register when adaption is initiated and will remain unchanged.

- Obtain a reasonably well-converged solution before performing an adaption. If you adapt to an incorrect solution, cells will be added in the wrong region of the flow.

  Use careful judgment in deciding how well to converge the solution before adapting, because there is a trade-off between adapting too early to an unconverged solution and wasting time by continuing to iterate when the solution is not changing significantly. This does not directly apply to dynamic adaption, because here the solution is adapted either at every iteration or at every time step, depending on which solver is being used.

- Write a case and data file before starting the adaption process. If you generate an undesirable mesh, you can restart the process with the saved files. This does not directly apply to dynamic adaption, because here the solution is adapted either at every iteration or at every time step, depending on which solver is being used.

- Select suitable variables when performing gradient adaption. For some flows, the choice is clear. For instance, adapting on gradients of pressure is a good criterion for refining in the region of shock waves. In most incompressible flows, however, it makes little sense to refine on pressure gradients. A more suitable parameter in an incompressible flow might be mean velocity gradients. If the flow feature of interest is a turbulent shear flow, it will be important to resolve the gradients of turbulent kinetic energy and turbulent energy dissipation, so these might be appropriate refinement variables. In reacting flows, temperature or concentration (or mole or mass fraction) of reacting species might be appropriate.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Do not over-refine a particular region of the solution domain. It causes very large gradients in cell volume. Such poor adaption practice can adversely affect the accuracy of the solution.

29.2. Boundary Adaption

This section describes how to perform boundary adaption. For more information, see Boundary Adaption in the Theory Guide.

For additional information, see the following section:
29.2.1. Performing Boundary Adaption

29.2.1. Performing Boundary Adaption

You can perform the boundary adaption in three different ways based on:

- number of cells

  In this case, the distance of a cell from the boundary is measured in number of cells.

- normal distance

  In this case, the cell refinement is based on the normal distance of a cell from the boundary.

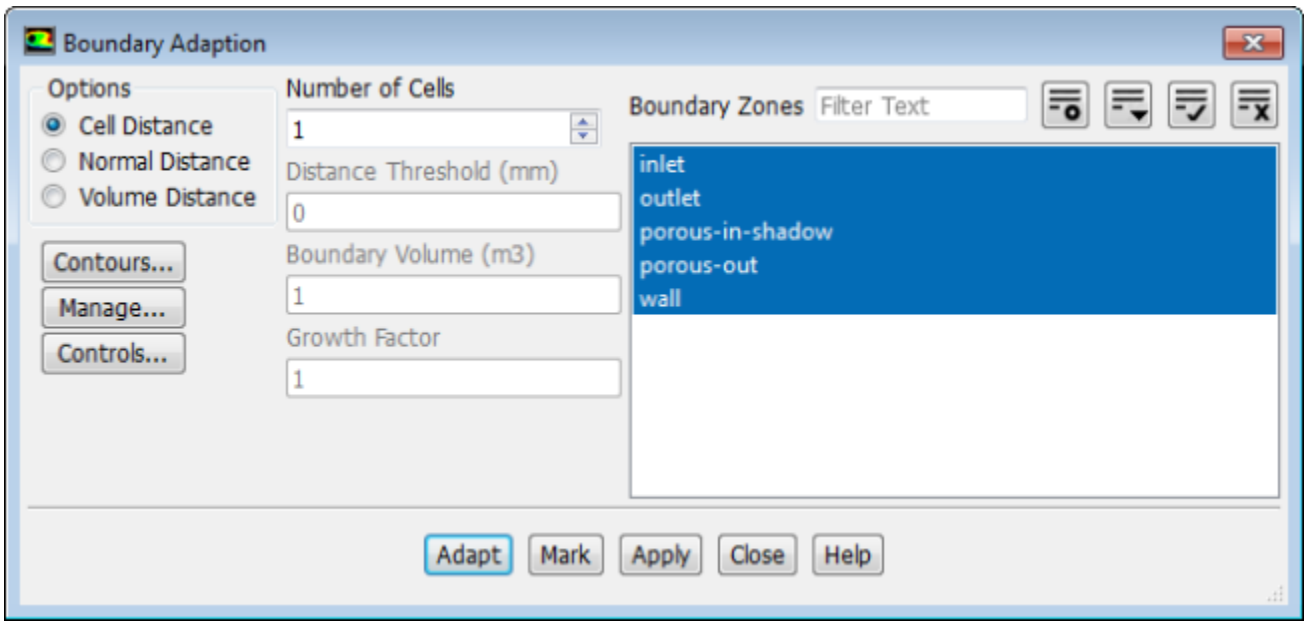
- target boundary volume

  In this case, the cell refinement is based on a target boundary volume and growth factor.

You can use any of these methods in the Boundary Adaption Dialog Box (p. 2673) (Figure 29.3: The Boundary Adaption Dialog Box (p. 1685)).

Setting Up Domain → Adapt → Mark/Adapt Cells → Boundary...

Figure 29.3: The Boundary Adaption Dialog Box



29.2.1.1. Boundary Adaption Based on Number of Cells

The procedure for performing adaption based on the distance of a cell from the boundary in terms of the number of cells is as follows:

1. In the Boundary Adaption Dialog Box (p. 2673) (Figure 29.3: The Boundary Adaption Dialog Box (p. 1685)), select **Cell Distance** under **Options**, choose the boundary zones near which you want to refine cells in the **Boundary Zones** list, and click **Apply**.

   This operation is performed to fill the cell distance variable for each cell to be visualized in Step 2.

2. (optional) Click the **Contours...** button to open the Contours Dialog Box (p. 2676).

   a. Enable **Filled** contours, disable **Node Values**.

   b. Select **Adaption...** and **Boundary Cell Distance** in the **Contours of** drop-down list.

   c. Select the appropriate surfaces (3D only).

   d. Click **Display** to see the location of cells with each value of boundary cell distance.

      By displaying different ranges of values (as described in Specifying the Range of Magnitudes Displayed (p. 1759)), you can determine the cell distance of the cells you want to adapt.

3. Set the **Number of Cells** to the desired value.

   - If you retain the default value of 1, only those cells that have edges (2D) or faces (3D) on the specified boundary zone(s) (that is, those cells with a boundary cell distance of 1) will be marked or adapted.
   - If you increase the value to 2, cells with a boundary cell distance of 2 will also be marked/adapted, and so on.

4. (optional) If you want to set any adaption options (described in Mesh Adaption Controls Dialog Box (p. 2730)), click the **Controls...** button to open the Mesh Adaption Controls Dialog Box (p. 2730).

5. Click **Mark** to mark the cells for refinement by placing them in an adaption register (which can be manipulated as described in Manipulating Adaption Registers (p. 1701)), or click **Adapt** to perform the refinement immediately.

29.2.1.2. Boundary Adaption Based on Normal Distance

The procedure for performing refinement based on a cell's normal distance from the boundary (that is, the distance between the centroid of a cell and the boundary) is as follows:

1. In the Boundary Adaption Dialog Box (p. 2673), select **Normal Distance** under **Options**, choose the boundary zones near which you want to refine cells in the **Boundary Zones** list, and click **Apply**.

   This operation is performed to fill the cell distance variable for each cell to be visualized in Step 2.

2. (optional) Open the Contours Dialog Box (p. 2676) by clicking on the **Contours...** button.

   a. Enable **Filled** contours, disable **Node Values**.

   b. Choose **Adaption...** and **Boundary Normal Distance** in the **Contours of** drop-down list.

   c. Select the appropriate surfaces (3D only).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

d. Click **Display** to see the location of cells with each value of normal distance.

   By displaying different ranges of values (as described in Specifying the Range of Magnitudes Displayed (p. 1759)), you can determine the normal distance of the cells you want to adapt.

3. Set the **Distance Threshold** to the desired value. Cells with a normal distance to the selected boundary zone(s) less than or equal to this value will be marked or adapted.

4. (optional) If you want to set any adaption options (described in Mesh Adaption Controls Dialog Box (p. 2730)), click the **Controls...** button to open the Mesh Adaption Controls Dialog Box (p. 2730).

5. Click **Mark** to mark the cells for refinement by placing them in an adaption register (which can be manipulated as described in Manipulating Adaption Registers (p. 1701)), or click **Adapt** to perform the refinement immediately.

29.2.1.3. Boundary Adaption Based on Target Boundary Volume

This boundary adaption allows you to produce exponentially larger (or smaller) cells as you get further from the boundaries. The cells are marked for refinement based on the following equation:

$$V_n > V_{boundary} e^{\alpha d} \tag{29.1}$$

where V_n is the cell volume, $V_{boundary}$ is the specified boundary volume (**Boundary Volume**), α is the exponential growth factor (**Growth Factor**), and d is the normal distance of the cell centroid from the selected boundaries. $V_{boundary} e^{\alpha d}$ is the target volume for a cell.

The procedure for this type of boundary refinement is as follows:

1. In the Boundary Adaption Dialog Box (p. 2673), select **Volume Distance** under **Options**, set the **Boundary Volume** and **Growth Factor** to the desired values, select the boundary zones in the **Boundary Zones** list where you want the **Boundary Volume** to be applied, and click **Apply**.

   This operation is performed to fill the cell distance variable for each cell to be visualized in Step 2.

2. (optional) Open the Contours Dialog Box (p. 2676) by clicking the **Contours...** button.

   a. Enable **Filled** contours, disable **Node Values**.

   b. Choose **Adaption...** and **Boundary Volume Distance** in the **Contours of** drop-down list.

   c. Select the appropriate surfaces (3D only).

   d. Click **Display** to see the contours of the target volume.

      By displaying different ranges of values (as described in Specifying the Range of Magnitudes Displayed (p. 1759)), you can determine the normal distance of the cells you want to adapt.

   You can modify the values of the inputs (**Boundary Volume**, **Growth Factor**, and/or **Boundary Zones**), click **Apply** in the **Boundary Adaption** dialog box, and then redisplay the contour plot to visualize the modified target volume distribution.

3. (optional) If you want to set any adaption options (described in Mesh Adaption Controls Dialog Box (p. 2730)), click the **Controls...** button to open the Mesh Adaption Controls Dialog Box (p. 2730).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click **Mark** to mark the cells for refinement by placing them in an adaption register (which can be manipulated as described in Manipulating Adaption Registers (p. 1701)), or click **Adapt** to perform the refinement immediately.

29.3. Gradient Adaption

This section describes how to perform gradient adaption. For more information, see Gradient Adaption in the Theory Guide.

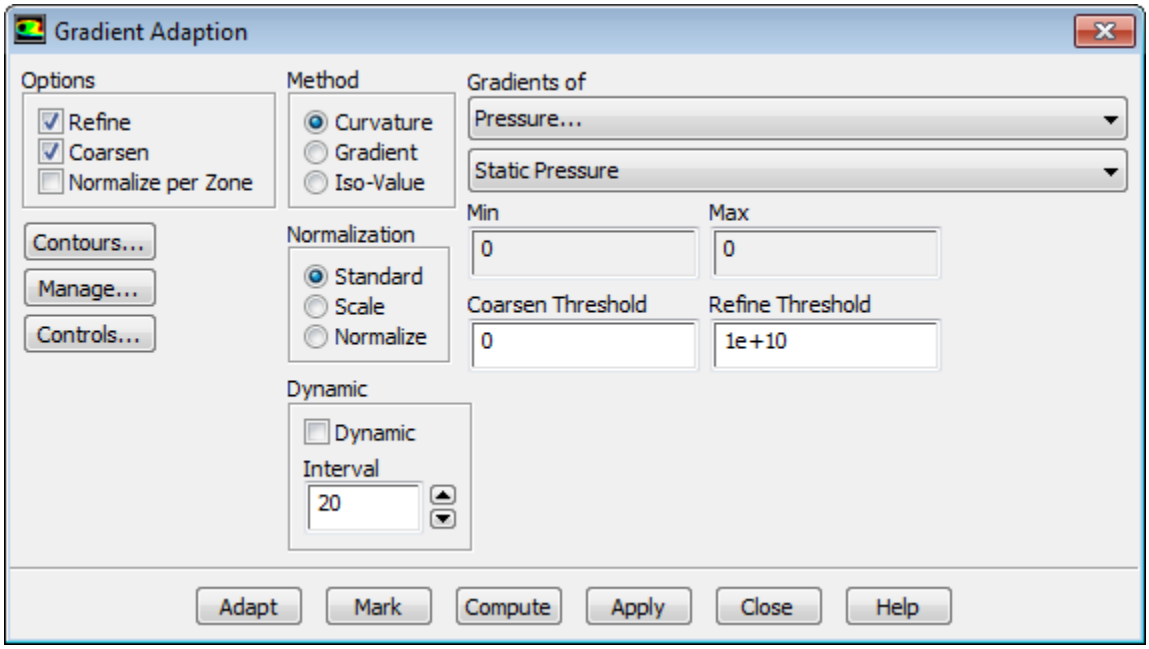
For additional information, see the following section:
29.3.1. Performing Gradient Adaption

29.3.1. Performing Gradient Adaption

The Gradient Adaption Dialog Box (p. 2710) (Figure 29.4: The Gradient Adaption Dialog Box (p. 1688)) allows you to perform gradient adaption.

Setting Up Domain → Adapt → Mark/Adapt Cells → Gradient...

Figure 29.4: The Gradient Adaption Dialog Box



The procedure for performing gradient adaption is as follows:

1. Select the appropriate adaption method.

   - **Curvature** is the default method, and is recommended for problems with smooth solutions.
   - **Gradient** is recommended for problems with strong shocks (for example, supersonic inviscid flows).
   - **Iso-Value** is recommended for problems where derivatives are not helpful, or when you want to customize the adaption criterion (using custom field functions, user-defined scalars, and so on).

2. Select a **Normalization** method:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Standard**, if normalization of the gradient or curvature is not to be performed.
- **Scale**, if the gradient or curvature is to be scaled by the average value in the domain.
- **Normalize**, if the gradient or curvature is to be scaled by the maximum value of the variable in the domain (that is, the gradient or curvature is bounded by [0, 1]).

Using either scaling or normalization makes the setting of the refine and coarsen thresholds much simpler, and almost independent of the current solution and specific problem.

This is especially important when using the automated dynamic adaption process.

3. Select the required solution variable in the **Gradients of** drop-down list.
4. Click **Compute**.
5. Click **Contours...** to open the Contours Dialog Box (p. 2676).

   a. Enable **Filled** contours, disable **Node Values**, and select **Adaption...** and **Existing Value** in the **Contours of** drop-down lists.

   b. Select the appropriate surfaces (3D only).

   c. Click **Display** to see the location of cells with each curvature value.

      By displaying different ranges of values (as described in Specifying the Range of Magnitudes Displayed (p. 1759)), you can determine the range of curvatures for which you want to adapt cells.

      If you are using normalization, the range for the curvatures of any variable will always be [0, 1].

6. Set the values for **Refine Threshold**.

   Cells with gradient values above this value will be either marked or refined.

7. Select the **Normalize per Zone** option for cases where different flow conditions exist for different zones.

   This approach of *zonal normalization* normalizes (that is, scales) each zone of the domain, in contrast to normalization on the whole domain. This approach is useful for dynamic adaption (see Dynamic Gradient Adaption (p. 1690) for details), where you want to solve the flow problem involving different flow intensities in the different cell zones.

   If you use gradient adaption for the whole domain, the small gradients may be neglected in comparison to large gradients depending on the adaption threshold. Enabling **Normalize per Zone** in the Gradient Adaption Dialog Box (p. 2710) will scale or normalize each zone independently, which means the strongest gradient for each zone is considered separately for adaption of that zone.

   Note

   If you expect gradients of different intensities throughout the domain and you want to resolve them, separate the domain into different zones for precise zonal normalization. This approach is referred as zonal adaption.

8. If you want to coarsen the mesh, set the **Coarsen Threshold** to a nonzero value. Cells with gradient values below the specified value will be either marked or coarsened.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. To set adaption options (described in Mesh Adaption Controls Dialog Box (p. 2730)), click **Controls...** to open the Mesh Adaption Controls Dialog Box (p. 2730).

10. To mark the cells for adaption (refinement/coarsening), click **Mark**. You can then place the cells in an adaption register, which can be manipulated (as described in Manipulating Adaption Registers (p. 1701)). To perform the adaption immediately, click **Adapt**.

Note

To disable refinement, coarsening, or marking for refinement/coarsening, disable the **Refine** or **Coarsen** option before marking or adapting.

29.4. Dynamic Gradient Adaption

This section describes how to perform dynamic gradient adaption. In contrast with the static gradient adaption (Gradient Adaption in the Theory Guide), dynamic gradient adaption is a fully automated process. For time dependent and for steady-state problems, you can perform the entire solution without changing the initial settings. That is, you can let the solver periodically perform adaptions without changing/entering any parameter.

Note

Dynamic gradient adaption based on custom field functions is not supported when running ANSYS Fluent in parallel.

For additional information, see the following section:
29.4.1. Dynamic Gradient Adaption Approach

29.4.1. Dynamic Gradient Adaption Approach

The dynamic gradient adaption executes the gradient adaption automatically. Though all options of gradient adaption are valid for the dynamic gradient adaption, some specific settings are recommended:

In the **Gradient Adaption** dialog box:

- Enable the **Refine** and **Coarsen** options.
- The **Normalize per Zone** enables zonal normalization for the dynamic adaption. See Performing Gradient Adaption (p. 1688) (step 7) for details.
- For **Normalization**, use either the **Scale** or the **Normalize** option.

  *The non-normalized values of the gradient or the curvature of a variable (obtained by selecting **Standard** for the **Normalization**) are generally strongly solution dependent, and therefore would require re-adjustment of the **Coarsen Threshold** and **Refine Threshold** as the solution proceeds.*

- For dynamic adaption, scaling is preferred if you want to resolve regions of small values of the gradient (or curvature/isovalues) accurately, in addition to the region of highest gradient (or curvature/isovalues).

  Scaling does not take very high values of the gradient or curvature into account to the degree that normalization does.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The starting values for **Refine Threshold** and **Coarsen Threshold** are 1e10 and 0 respectively.

  The more refinement you want, the smaller these values should be.

- Specify the **Interval** between two consecutive automatic mesh adaptions. Depending on whether you are performing a steady-state or a time-dependent solution, specify **Interval** in iterations or time steps, respectively.

  This value depends on the type of problem solved and the time step used (where applicable). For steady-state problems, values of 100 or higher are reasonable. For time-dependent problems, values of 10 or lower are often required.

  If you are using the density-based explicit solver with explicit transient formulation, your input will be in number of iterations.

In the **Mesh Adaption Controls** dialog box:

- Set values for **Min # of Cells**, **Max # of Cells**, **Max Level of Refine**, or **Min Cell Volume**.

  The limits for the **Min # of Cells** and **Max # of Cells** can affect the **Coarsen Threshold** and **Refine Threshold** values. If either the **Min # of Cells** or the **Max # of Cells** are violated, the **Coarsen Threshold** or the **Refine Threshold** are adjusted to fulfill the limits for the **Min # of Cells** or the **Max # of Cells**.

- The default value for **Max Level of Refine** is 2, which is a good start for most problems. If required, you can increase this value.

  Important

  Even in a 2D problem, the default value of 2 can increase the number of cells by a factor of 16 in the adapted regions. A value of zero leaves this parameter unbounded: in this case, you should use a suitable limit for **Min Cell Volume**.

29.4.1.1. Examples of Dynamic Gradient Adaption

Example 1: Steady-state problem

Consider a supersonic flow over a blunt body. To determine the wave drag for such problem, first resolve the shock wave. Start with a coarse mesh and set up dynamic adaption. As you start iterating the solution, the solver will produce a blurred shock, probably in an incorrect location. After the adaptions, the shock will become sharper and move into the correct location.

Example 2: Time-dependent problem

Consider a traveling shock wave. To determine the precise pressure amplitudes and arrival times at a number of locations, you must resolve the shock wave over the time, so that you can maintain the correct shock strength and its location. Dynamic adaption is efficient in this case, as it refines the mesh near the shock, and at the same time, it coarsens the mesh wherever needed.

29.5. Isovalue Adaption

This section describes how to perform isovalue adaption. For more information, see Isovalue Adaption in the Theory Guide.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

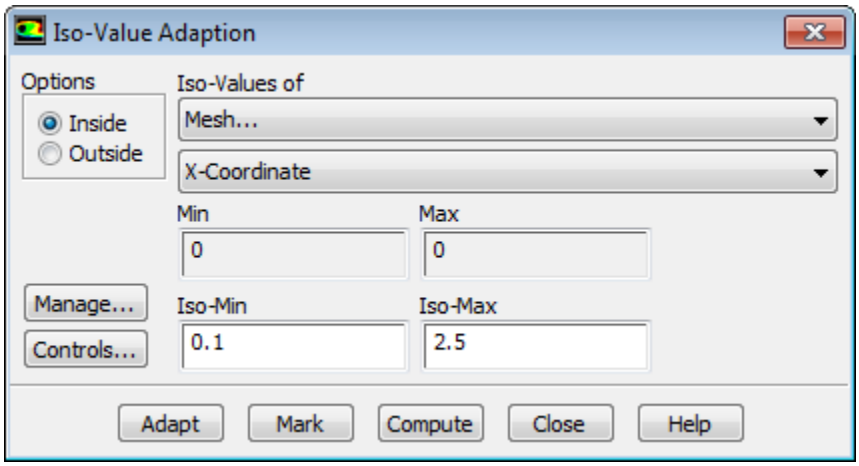
For additional information, see the following section:
29.5.1. Performing Isovalue Adaption

29.5.1. Performing Isovalue Adaption

You can perform isovalue adaption in the Iso-Value Adaption Dialog Box (p. 2722) (Figure 29.5: The Iso-Value Adaption Dialog Box (p. 1692)).

Setting Up Domain → Adapt → Mark/Adapt Cells → Iso-Value...

Figure 29.5: The Iso-Value Adaption Dialog Box



The general procedure for performing isovalue adaption is as follows:

1. Select the desired solution variable in the **Iso-Values of** drop-down lists and click **Compute** to update the **Min** and **Max** fields.
2. Choose the **Inside** or **Outside** option and set the **Iso-Min** and **Iso-Max** values.
   - If you choose **Inside**, cells with isovalues between **Iso-Min** and **Iso-Max** will be marked or refined.
   - If you choose **Outside**, cells with isovalues less than **Iso-Min** or greater than **Iso-Max** will be marked or refined.
3. (optional) If you want to set any adaption options (described in Mesh Adaption Controls Dialog Box (p. 2730)), click the **Controls...** button to open the Mesh Adaption Controls Dialog Box (p. 2730).
4. Click **Mark** to mark the cells for refinement by placing them in an adaption register (which can be manipulated as described in Manipulating Adaption Registers (p. 1701)), or click **Adapt** to perform the refinement immediately.

29.6. Region Adaption

This section describes how to perform region adaption. For more information, see Region Adaption in the Theory Guide.

For additional information, see the following section:
29.6.1. Performing Region Adaption

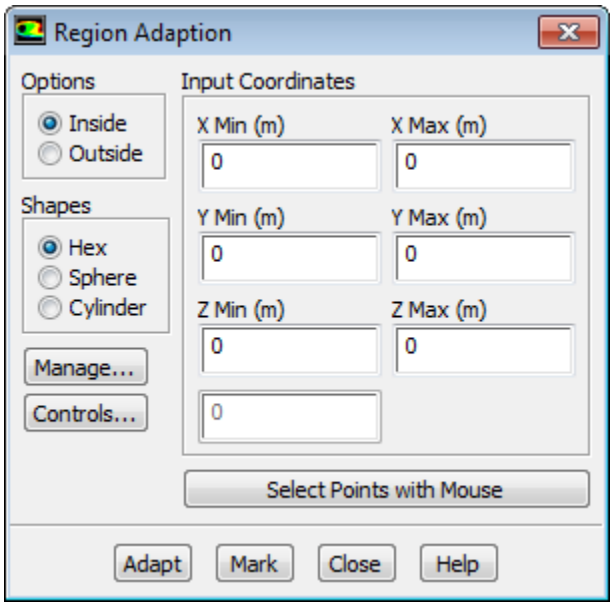
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

29.6.1. Performing Region Adaption

You will perform region adaption in the Region Adaption Dialog Box (p. 2782) (Figure 29.6: The Region Adaption Dialog Box (p. 1693)).

Setting Up Domain → Adapt → Mark/Adapt Cells → Region...

Figure 29.6: The Region Adaption Dialog Box



The procedure for performing region adaption is as follows:

1. In the Region Adaption Dialog Box (p. 2782), select the **Inside** or **Outside** option.

   - If you choose **Inside**, cells with centroids within the specified region will be marked or refined.
   - If you choose **Outside**, cells with centroids outside the specified region will be marked or refined.

2. Specify the shape of the region.

   In 2D, you may specify a **Quad** (that is, a quadrilateral), **Circle**, or **Cylinder** by making a selection from the **Shape** group box. In 3D, your options include **Hex** (that is, a hexahedron), **Sphere**, or **Cylinder**.

3. Define the region by entering values into the dialog box or by using the mouse.

   In the dialog box, the inputs are as follows:

   - To define a hexahedron or quadrilateral, enter the coordinates of two points defining the diagonal of the box.

     For a hexahedron, define **X Min**, **Y Min**, and **Z Min**, as well as **X Max**, **Y Max**, and **Z Max**. For a quadrilateral, define **X Min** and **Y Min**, as well as **X Max** and **Y Max**.

   - To define a sphere or circle, enter the values for the **Radius** and the coordinates of its center: **X Center**, **Y Center**, and **Z Center** for a sphere, or **X Center** and **Y Center** for a circle.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To define a cylinder, enter the value for the **Radius** and the minimum and maximum coordinates defining the cylinder axis: **X-Axis Min**, **Y-Axis Min**, and **Z-Axis Min**, as well as **X-Axis Max**, **Y-Axis Max**, and **Z-Axis Max** for 3D, or **X-Axis Min** and **Y-Axis Min**, as well as **X-Axis Max** and **Y-Axis Max** for 2D. In 2D, this will be the width of the resulting rectangle.

4. To define the region using the mouse, click the **Select Points with Mouse** button. Using the right mouse button, select the input coordinates from a display of the mesh or solution field. After selecting the points, the values will be loaded automatically into the appropriate fields in the dialog box. See Controlling the Mouse Button Functions (p. 1800) for details about mouse button functions.

   You have the option of editing these values before marking or adapting.

   - To define a hexahedron or quadrilateral, select the two points of the diagonal in any order.
   - To define a sphere or circle, first select the location of the centroid and then select a point that lies on the sphere/circle (that is, a point that is one radius away from the centroid).
   - To define a cylinder, first select the two points that define the cylinder axis and then select a point that is one radius away from the axis.

5. (optional) If you want to set any adaption options (described in Mesh Adaption Controls Dialog Box (p. 2730)), click the **Controls...** button to open the Mesh Adaption Controls Dialog Box (p. 2730).

6. Click **Mark** to mark the cells for refinement by placing them in an adaption register (which can be manipulated as described in Manipulating Adaption Registers (p. 1701)), or click **Adapt** to perform the refinement immediately.

29.7. Volume Adaption

This section describes how to perform volume adaption. For more information, see Volume Adaption in the Theory Guide.

For additional information, see the following section:
29.7.1. Performing Volume Adaption

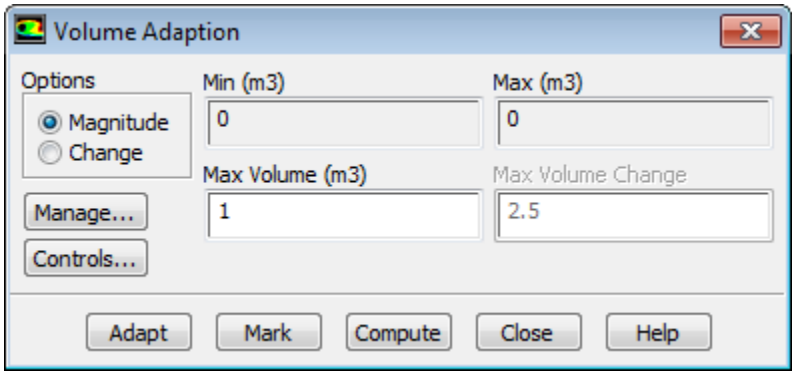
29.7.1. Performing Volume Adaption

You will perform volume adaption in the Volume Adaption Dialog Box (p. 2838) (Figure 29.7: The Volume Adaption Dialog Box (p. 1695)).

Setting Up Domain → Adapt → Mark/Adapt Cells → Volume...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 29.7: The Volume Adaption Dialog Box



The procedure for performing volume adaption is as follows:

1. In the Volume Adaption Dialog Box (p. 2838), specify whether you want to adapt based on volume magnitude or volume change by selecting the **Magnitude** or **Change** option.

2. Click **Compute** to update the **Min** and **Max** fields. These fields will show the range of cell volumes or cell volume changes (defined in Volume Adaption Approach in the Theory Guide), depending on your selection in step 1.

3. Set the **Max Volume** or **Max Volume Change** value.

   a. If you have chosen to adapt based on volume **Magnitude**, cells that have volumes greater than **Max Volume** will be marked or refined.

   b. If you are adapting based on volume **Change**, cells with volume changes greater than **Max Volume Change** will be marked or refined.

4. (optional) If you want to set any adaption options (described in Mesh Adaption Controls Dialog Box (p. 2730)), click the **Controls...** button to open the Mesh Adaption Controls Dialog Box (p. 2730).

5. Click **Mark** to mark the cells for refinement by placing them in an adaption register (which can be manipulated, as described in Manipulating Adaption Registers (p. 1701)), or click **Adapt** to perform the refinement immediately.

29.8. Yplus/Ystar Adaption

This section describes how to perform Yplus/Ystar adaption. For more information, see Yplus/Ystar Adaption in the Theory Guide.

For additional information, see the following section:
29.8.1. Performing Yplus or Ystar Adaption

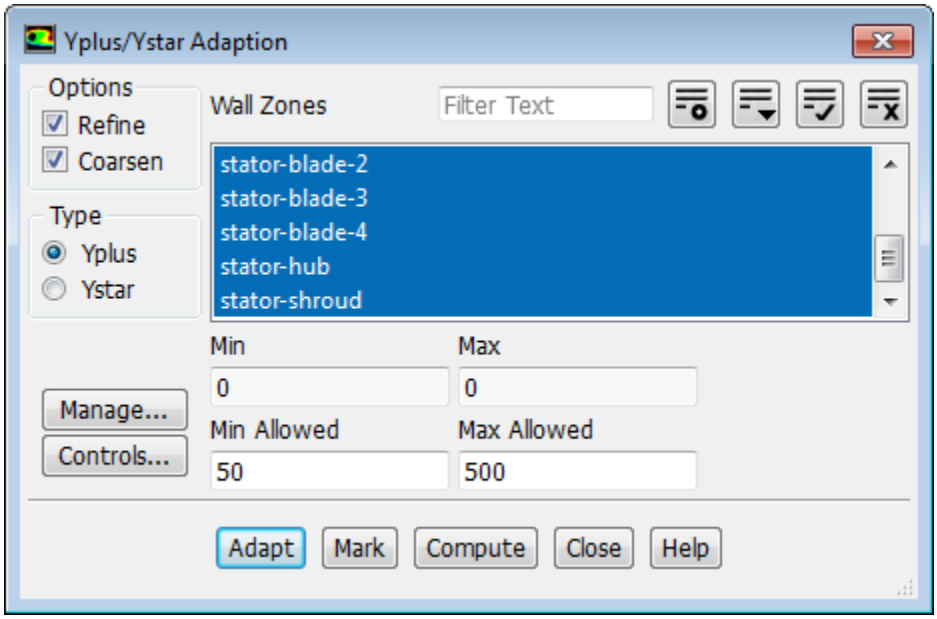
29.8.1. Performing Yplus or Ystar Adaption

You will perform Yplus or Ystar adaption in the Yplus/Ystar Adaption Dialog Box (p. 2842) (Figure 29.8: The Yplus/Ystar Adaption Dialog Box (p. 1696)).

Setting Up Domain → Adapt → Mark/Adapt Cells → Yplus/Ystar...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 29.8: The Yplus/Ystar Adaption Dialog Box



The procedure for performing y^+ or y^* adaption is as follows:

1. In the Yplus/Ystar Adaption Dialog Box (p. 2842), select **Yplus** or **Ystar** as the adaption **Type**.
   - Select **Yplus** if you are using a y^+-insensitive wall treatment.
   - If you are using wall functions, you can select either type.
2. Select the wall zones for which you want boundary cells to be marked or adapted in the **Wall Zones** list, and click **Compute** to update the **Min** and **Max** fields. The values displayed are the minimum and maximum values for all wall zones, not just of those selected.
3. Set the **Min Allowed** and **Max Allowed**. Cells with y^+ or y^* values below **Min Allowed** will be coarsened or marked for coarsening, and cells with y^+ or y^* values above **Max Allowed** will be refined or marked for refinement.
4. (optional) If you want to set any adaption options (described in Mesh Adaption Controls Dialog Box (p. 2730)), click the **Controls...** button to open the Mesh Adaption Controls Dialog Box (p. 2730).
5. Click **Mark** to mark the cells for adaption (refinement/coarsening) by placing them in an adaption register (which can be manipulated as described in Manipulating Adaption Registers (p. 1701)), or click **Adapt** to perform the adaption immediately.

To disable refinement or coarsening, or marking for refinement or coarsening, turn off the **Refine** or **Coarsen** option before marking or adapting.

Note

y^+ and y^* values are only available for cells with faces on the wall.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

29.9. Anisotropic Adaption

This section describes how to perform anisotropic adaption. For more information, see Anisotropic Adaption in the Theory Guide.

For additional information, see the following sections:

29.9.1. Limitations of Anisotropic Adaption
29.9.2. Performing Anisotropic Adaption
29.9.3. Boundary Layer Redistribution

29.9.1. Limitations of Anisotropic Adaption

Since anisotropic adaption is available only for specific cell types, the following limitations exist:

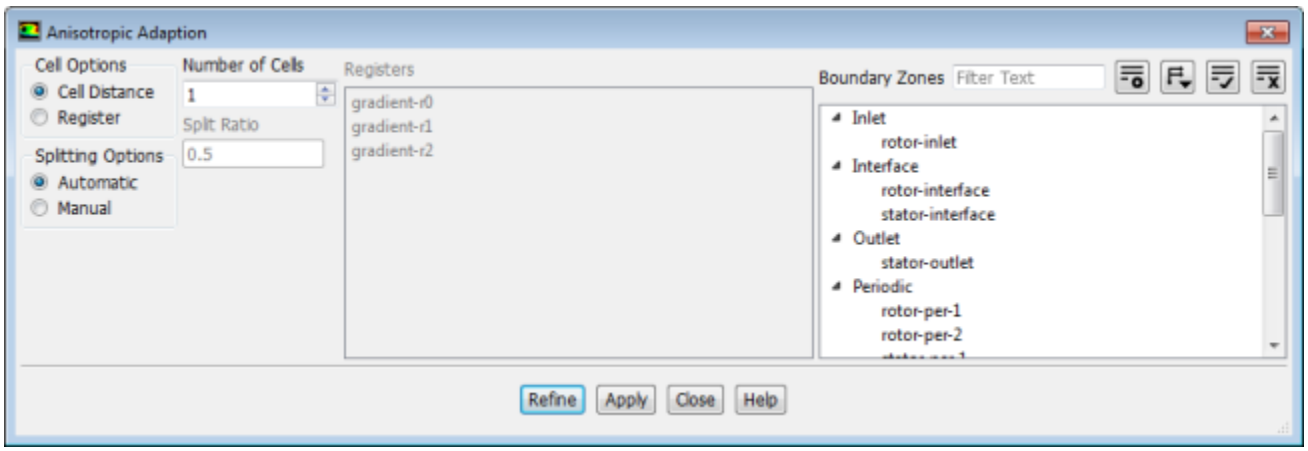
- It is only available in 3D.
- It only works for hexahedral cells or prism cells.
- Each cell can only be split into two, with a given splitting ratio in the normal direction of the boundary face. Multiple layers can be achieved by multiple refinement.
- Each cell to be split can only be reached once from any of the boundary faces; otherwise, the refinement will not be processed.
- Unlike other adaption functionalities, the subdivided cells cannot be coarsened again, because all the cells that are adjacent to the refined cells are converted into polyhedral cells.

29.9.2. Performing Anisotropic Adaption

You will perform anisotropic adaption in the Anisotropic Adaption Dialog Box (p. 2671) (Figure 29.9: The Anisotropic Adaption Dialog Box (p. 1697)).

Setting Up Domain → Adapt → Mark/Adapt Cells → Anisotropic...

Figure 29.9: The Anisotropic Adaption Dialog Box



The procedure for performing anisotropic adaption is as follows:

1. In the Anisotropic Adaption Dialog Box (p. 2671), select **Cell Distance** or **Register** from the **Cell Options** group box. This allows you to control the marking of boundary layer cells. Select **Cell Distance** and enter

the **Number of Cells** to be adapted and marked using the distance from the boundary zone. Select **Register** to adapt and mark cells using an existing **Register**.

2. The **Splitting Options** control how the splitting ratio is computed. It is defined as follows:

$$\text{splitting ratio} = \frac{\text{height of the splitting point to the base face}}{\text{original height of the cell}}$$

By default, **Automatic** is selected, resulting in the ratio being computed automatically from the mesh. If you selected **Manual**, enter the desired **Split Ratio**. If you choose to compute the split ratio automatically, the split ratio of the first layer is computed, and it may be 0.5 if the original cells are uniformly distributed, resulting in the height of the first layer being the same as the height of the second layer.

Important

Note that the **Split Ratio** that you enter is only applicable to the first layer, and all the other layers are split with a ratio of 0.5.

---

3. For the **Cell Distance** option, one or more boundary face zones must be selected before marking the cells for refinement.

4. For the **Register** option, select a register in the list. Note that one or more boundary face zones must be selected before doing the refinement.

5. Click **Refine** to refine the marked cells.

29.9.3. Boundary Layer Redistribution

After performing anisotropic adaption the boundary layer zone will generally not satisfy the original growth rate. To restore the desired growth rate you can use the `mesh/redistribute-boundary-layer` TUI command. You will be prompted to enter a face zone and the desired growth rate. The face zone you specify must be adjacent to the boundary layer zone that you want to redistribute.

```
>  mesh redistribute-boundary-layer
face zone id/name [] wall-fluid
growth rate [1.1] 1.2
```

Fluent will redistribute the nodes in a boundary region that extends into the mesh from the face zone you specify until one of the following criteria is encountered:

- the edge of the domain
- a change in zone
- a change in element type

As with anisotropic adaption itself, boundary layer redistribution is only available in 3D problems.

29.10. Geometry-Based Adaption

This section describes how to perform geometry-based adaption. For more information, see Geometry-Based Adaption in the Theory Guide.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

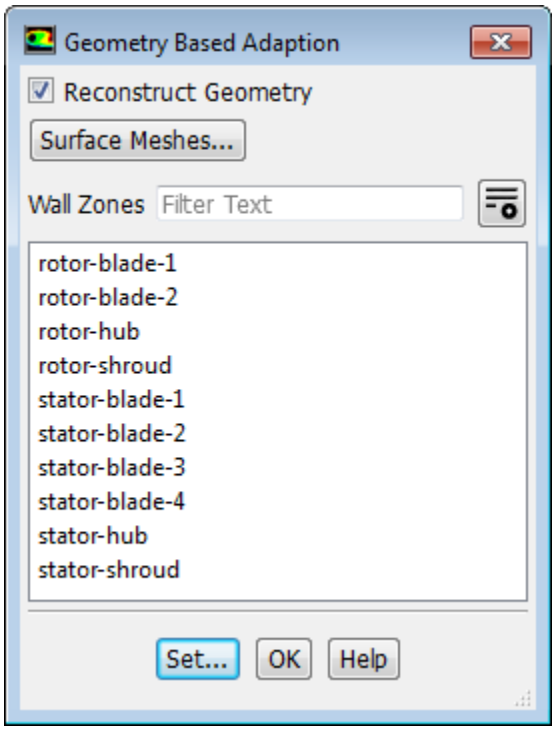
For additional information, see the following section:
29.10.1. Performing Geometry-Based Adaption

29.10.1. Performing Geometry-Based Adaption

The Geometry Based Adaption Dialog Box (p. 2709) (Figure 29.10: The Geometry Based Adaption Dialog Box (p. 1699)) allows you to reconstruct the geometry while performing boundary adaption.

Setting Up Domain → Adapt → More → Geometry...

Figure 29.10: The Geometry Based Adaption Dialog Box

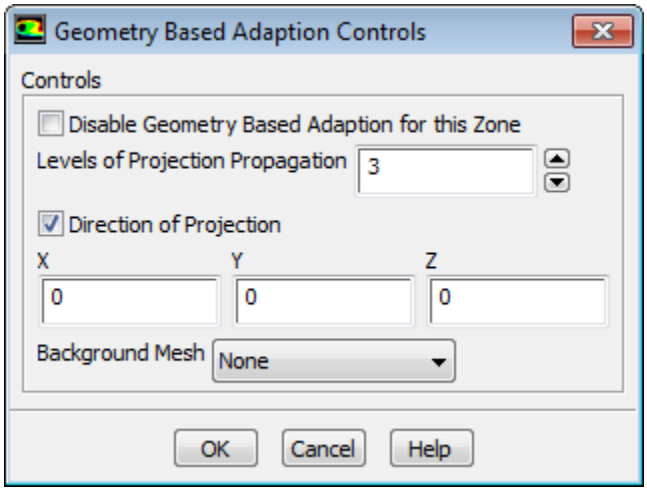


The procedure for performing geometry-based adaption is as follows:

1. Enable the **Reconstruct Geometry** option.
2. Under **Wall Zones**, select the zone you want to adapt and click **Set...**. The Geometry Based Adaption Controls Dialog Box (p. 2709) will open.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 29.11: The Geometry Based Adaption Controls Dialog Box



In the Geometry Based Adaption Controls Dialog Box (p. 2709), set the following parameters:

- Specify **Levels of Projection Propagation** to indicate the number of layers of the nodes you want to project.
- Enable **Direction of Projection** and specify the directions in which you want to project the nodes.

  This will make the **X** *,* **Y** *, and* **Z** *fields available. If you want node projection in the X direction, enter* 1 *for* **X** *. If you do not enable this option, the node projection will take place at the nearest point.*
- (optional) If you have a fine surface mesh for the geometry, you can use the **Background Mesh** option to load the surface mesh as a background mesh. This will project the nodes based on the background mesh and reconstruct the geometry more accurately.
- To disable the geometry reconstruction for any zone in the domain, enable the **Disable Geometry Based Adaption for this Zone** option.

3. To disable geometry-based adaption for the whole domain, disable **Reconstruct Geometry**.

   After setting the parameters for geometry-based adaption, proceed to perform mesh adaption.

29.11. Registers

This section describes how to use registers for adaption. For more information, see Registers in the Theory Guide.

Note

Similar to adaption registers, you can create cell registers, which are saved with the case file. For additional information on cell registers, see Creating Surfaces and Cell Registers for Displaying and Reporting Data (p. 1709) and Using Cell Registers (p. 1736).

For additional information, see the following sections:

29.11.1. Manipulating Adaption Registers
29.11.2. Modifying Adaption Marks
29.11.3. Displaying Registers

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

29.11.4. Adapting to Registers

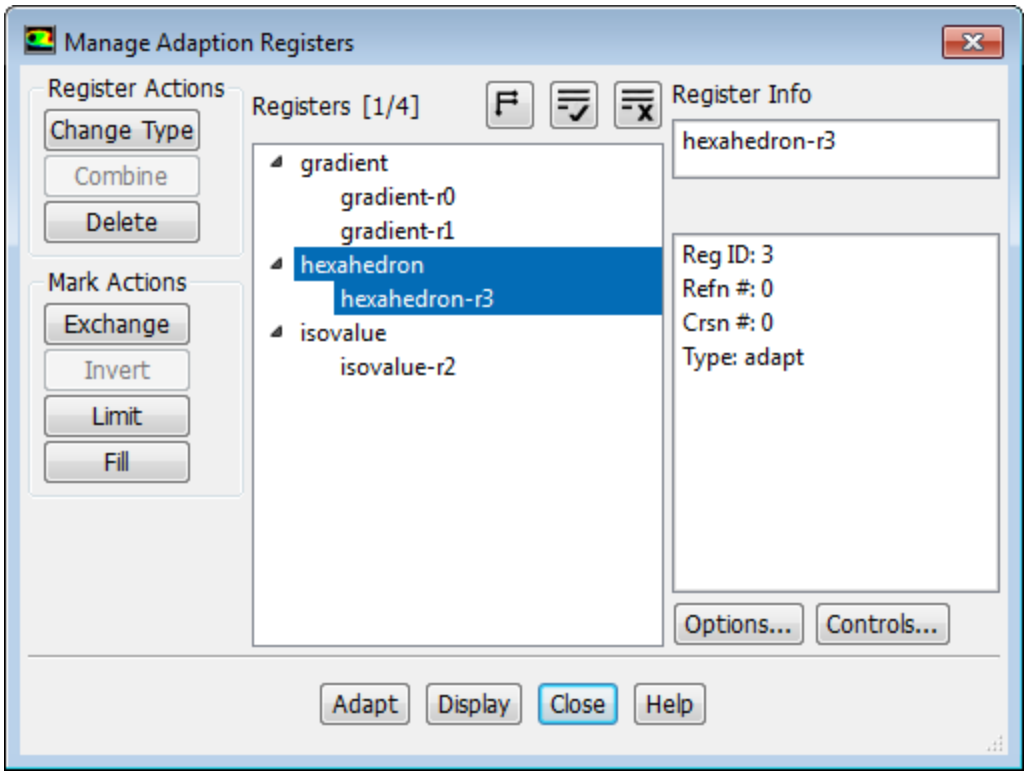
29.11.1. Manipulating Adaption Registers

You can manipulate, delete, and display adaption registers by marking cells for adaption. Since these registers are used to adapt the mesh, the ability to manipulate them provides additional control over the adaption process.

Management of adaption registers is performed in the Manage Adaption Registers Dialog Box (p. 2728) (Figure 29.12: The Manage Adaption Registers Dialog Box (p. 1701)). You can also open this dialog box by clicking on the **Manage...** button in any of the adaption dialog boxes.

Setting Up Domain → Adapt → Manage Registers...

Figure 29.12: The Manage Adaption Registers Dialog Box



You can modify and manipulate adaption registers by:

- changing the register types
- combining the registers
- deleting the registers

29.11.1.1. Changing Register Types

If the adaption register is converted to a mask, the cells marked for refinement are ACTIVE, and all other cells are INACTIVE (that is, the cells marked for coarsening are ignored). Generally, the adaption registers converted to masks are those that are generated by adaption functions that mark cells exclusively for refinement, such as region or isovalue adaption functions. The other major difference between adaption and mask registers is the manner in which they are combined.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To change the type of one or more registers from adaption to mask, or vice versa, do the following:

1. Choose the register(s) in the **Registers** list.

2. Click the **Change Type** button under **Register Actions**.

The new type of the register (if multiple registers are selected, the most recently selected or deselected register) will be shown as the **Type** under **Register Info**. Select each register individually to see what its current type is.

29.11.1.2. Combining Registers

After the individual adaption registers have been created and appropriately modified, they are combined to create hybrid adaption functions.

1. Any number of registers can be combined in the following manner:

   - All adaption registers are combined into a new adaption register.

   - All mask registers are combined into a new mask register.

   - The new adaption and mask registers are combined.

2. Any number of adaption registers can be combined in the following manner:

   - If the cell is marked for refinement in any of the registers, mark the cell for refinement in the new register (bitwise OR).

   - If the cell is marked for coarsening in all of the registers, mark the cell for coarsening in the new register (bitwise AND).

3. The mask registers are combined in a manner similar to the refinement marks. If any cell is marked ACTIVE, the cell in the new register is marked ACTIVE (bitwise OR).

4. Finally, in the combination of an adaption and mask register, only cells that are marked in the mask register can have an adaption mark in the combined register (bitwise AND).

For example, creating an adaption function based on pressure gradient may generate cells marked for refinement and coarsening throughout the entire solution domain. If this register is then combined with a mask register created from cells marked inside a sphere, only the cells inside the sphere will be marked for refinement or coarsening in the new register.

Note

The effect of masks depends on the order in which they are applied.

For example, consider two adjacent, circular masks. Applying one mask to the adaption register and then applying the other mask to the result of the first combination would give a much different result than applying the combination of the two masks to the initial adaption register. The second combination results in a greater possible number of marked cells.

To combine two or more registers, do the following:

1. Choose the registers in the **Registers** list.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click the **Combine** button under **Register Actions**.

The selected registers will remain intact, and the register(s) resulting from the combination will be added to the **Registers** list. In some instances, three new registers may be created:

- a combination of the adaption registers
- a combination of the mask registers
- a combination of the two combined registers

For more information about combining registers, see Adaption Registers in the Theory Guide.

29.11.1.3. Deleting Registers

The primary reason for deleting registers is to discard unwanted adaption registers. This will reduce confusion and the possibility of generating undesired results by selecting these discarded registers. In addition, only 32 adaption registers can exist at one time. Therefore, you should discard unwanted registers to make room for new ones. You can delete any number of adaption registers.

To permanently delete one or more registers, do the following:

1. Choose the register(s) in the **Registers** list.
2. Click the **Delete** button under **Register Actions**.

29.11.2. Modifying Adaption Marks

The adaption marks are the identifiers that designate whether a cell should be refined, coarsened, or neutral. The operations used for modifying the adaption marks are:

- **Exchange**: This changes the cells marked for refinement into cells marked for coarsening, and all cells originally marked for coarsening into cells marked for refinement. This operation is applied to adaption registers that have only refinement marks.

  For example, the exchange operation can be used to coarsen a rectangular region. First, create an adaption register that marks a rectangular region of cells for refinement. Then use the **Exchange** operation to modify the cell marks, creating a rectangular region with cells marked for coarsening.

- **Invert**: This operation can only be used with mask registers. It toggles the mask markings, that is, all cells marked ACTIVE are switched to INACTIVE, and all cells marked INACTIVE are switched to ACTIVE.

  For example, if you generate a mask that defines a circular region, you can quickly modify the mask to define the region outside of the circle using the **Invert** operation.

- **Limit**: This operation applies the present adaption volume limit to the selected adaption register. For information on adaption limits, see Mesh Adaption Controls (p. 1705). You generally use this operation to determine the effect of the present limits on the adaption process. You can use the volume limit to create a uniform mesh by setting the limit to refine only the large cells. After all the cells have reached a uniform size, you can continue the refinement process to the desired resolution.

- **Fill**: This operation marks the cells in the adaption register that are not marked for refinement. You can use the **Fill** operation to combine multiple registers to make a new register.

  Note the following:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– When you combine registers, a cell will be marked for coarsening only if it is marked for coarsening in all of the registers.

– If you create an adaption register with an operation that only marks cells for refinement, but you do not want to prohibit coarsening, use the **Fill** operation before combining the register with any other registers.

The process for modifying adaption marks is as follows:

1. Choose the register(s) in the **Registers** list.

2. Click the **Exchange**, **Invert**, **Limit**, or **Fill** button under **Mark Actions**.

29.11.3. Displaying Registers

Viewing the cell markings is often helpful in the process of creating hybrid adaption functions. You can plot a marker at the cell centroid and/or a wireframe of the cell to view the state of the cell. By default, the cells marked for refinement are colored in red, and the cells marked for coarsening are marked in cyan. In addition, cells marked ACTIVE in a mask register are also colored red. These are the cells that are marked for adaption, but the final number of cells added or subtracted from the mesh depends on the adaption limits and the mesh characteristics.

To display a register, do the following:

1. Choose the register in the **Registers** list.

2. Set the display options by clicking on the **Options...** button.

3. Click the **Display** button.

29.11.3.1. Adaption Display Options

Various aspects of the adaption register display can be modified, such as the wireframe visibility and shading, marker visibility, color, size, and symbol. Also, you can select either surface or zone meshes for the display.

The adaption register display capability allows you to view the cells that are flagged for adaption.

- Depending on the dimension of the problem and the number of flagged cells, you can customize the adaption display options. The most common method for viewing flagged cells in 2D is to draw the mesh and filled wireframes, but this is impractical in 3D. In three dimensions, you can plot the centroid markers of the cells with the mesh of selected boundary zones.

- You can use markers and/or wireframes to display the flagged cells in an adaption or mask register. The marker is a symbol placed at the centroid of the cell. There is a refine marker and a coarsen marker. You can change the symbol, color, and size of these markers. A wireframe is composed of the edges of the triangle or tetrahedron. Its color is the same as the respective marker color, and can be filled, if required.

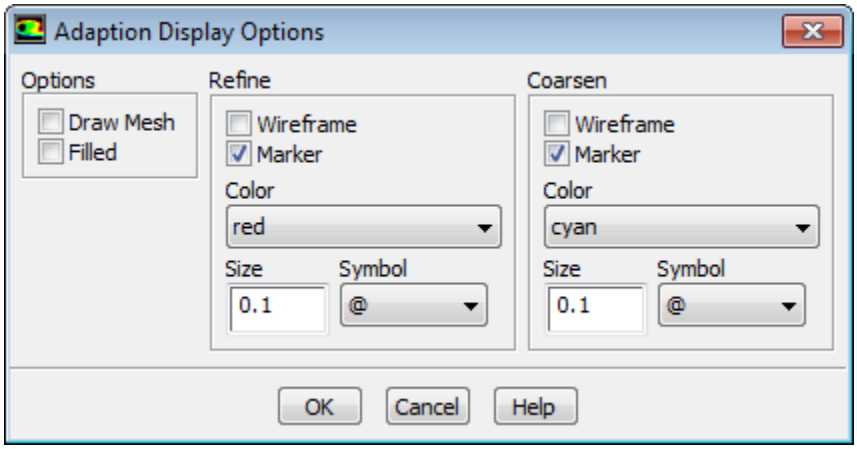
- Portions of the mesh can be drawn with the marker symbols or wireframes to aid in evaluating the location of marked cells.

All of these options are set in the Adaption Display Options Dialog Box (p. 2663) (Figure 29.13: The Adaption Display Options Dialog Box (p. 1705)). You can also open this dialog box by clicking on the **Options...** button in the Manage Adaption Registers Dialog Box (p. 2728).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Up Domain → Adapt → More → Display Options...

Figure 29.13: The Adaption Display Options Dialog Box



- To enable or disable the display of wireframes for cells marked for refinement/coarsening, turn the **Wireframe** option on or off under **Refine** and/or **Coarsen**. To draw filled wireframes (that is, using a solid color, instead of the outline) turn on the **Filled** option.
- To enable or disable the display of markers for cells marked for refinement/coarsening, turn the **Marker** option on or off under **Refine** and/or **Coarsen**. Use markers to specify their size in the **Size** field, and their symbol in the **Symbol** drop-down list.
- To change the color of the refine or coarsen markers/wireframes, select a new color in the **Color** drop-down list under **Refine** or **Coarsen**. By default, refine markers/wireframes are red and coarsen markers/wireframes are cyan.
- To include portions of the mesh in the register display, enable the **Draw Mesh** option. The Mesh Display Dialog Box (p. 2163) will open automatically, where you can set the mesh display parameters. When you click **Display** in the Manage Adaption Registers Dialog Box (p. 2728), the mesh display, as defined in the Mesh Display Dialog Box (p. 2163), will be included in the register display.

29.11.4. Adapting to Registers

These register tools provide you with the ability to create hybrid adaption functions customized to your flow-field application. The customized adaption function is used to direct the refinement and coarsening of the mesh.

To perform the adaption, follow these steps:

1. Choose the register in the **Registers** list.
2. Click the **Adapt** button.

29.12. Mesh Adaption Controls

ANSYS Fluent allows you to:

- Place restrictions on the cell zones.
- Limit adaption by cell volume or volume weight.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

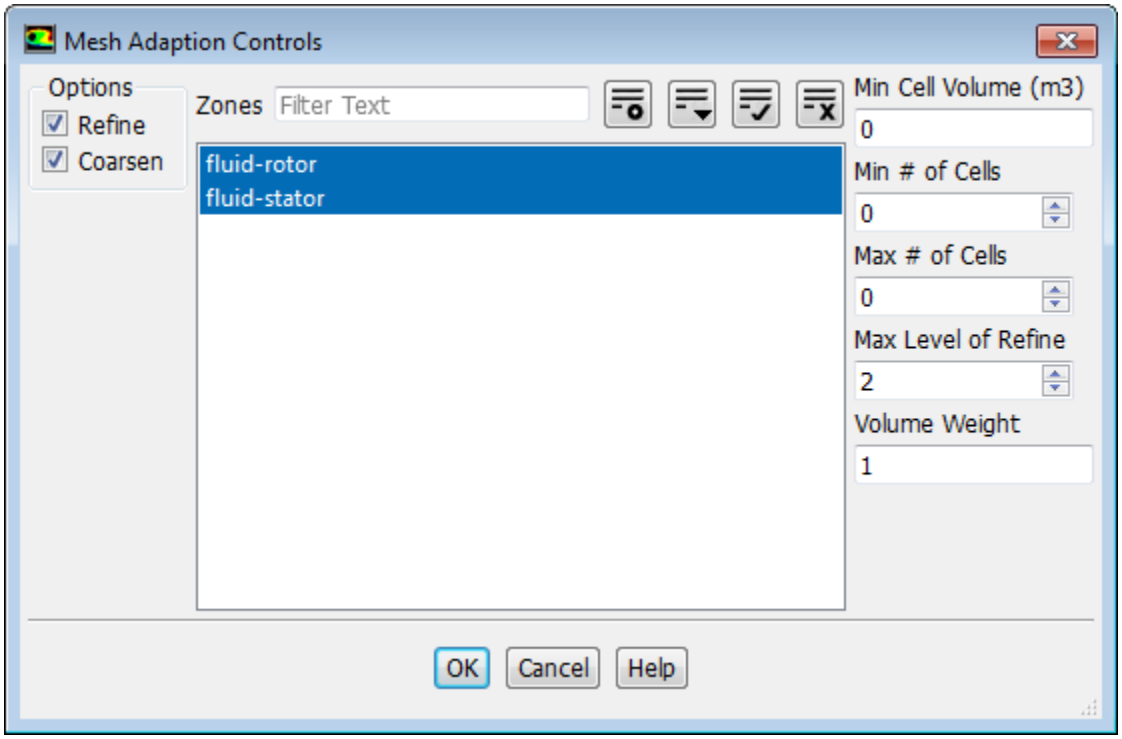
- Limit the total number of cells that can be produced from the adaption process.
- Modify the intensity of the volume weighting in the gradient function.
- Restrict the adaption process to refinement and/or coarsening, and control which nodes are eligible for possible elimination from the mesh during coarsening.

The parameters controlling the aspects of adaption are set in the Mesh Adaption Controls Dialog Box (p. 2730) (Figure 29.14: The Mesh Adaption Controls Dialog Box (p. 1706)).

You can open this dialog box by selecting **Controls...** from the **More** drop-down list in the **Setting Up Domain** tab (**Adapt** group box).

Setting Up Domain → Adapt → More → Controls...

Figure 29.14: The Mesh Adaption Controls Dialog Box



Note

Write a case and data file before starting the adaption process. Then, if you generate an undesirable mesh, you can restart the process with the saved files.

For additional information, see the following sections:

29.12.1. Limiting Adaption by Zone
29.12.2. Limiting Adaption by Cell Volume or Volume Weight
29.12.3. Limiting the Total Number of Cells
29.12.4. Controlling the Levels of Refinement During Hanging Node Adaption

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

29.12.1. Limiting Adaption by Zone

You can limit the adaption process to specified cell zones. The cells composing the fluid and solid regions of the analysis generally have very different resolution requirements and error indicators. Limiting the adaption to a specific cell zone and use different adaption functions to create the optimal mesh.

To limit the adaption to a particular cell zone (or to particular cell zones), select the cell zones in which you want to perform adaption in the **Zones** list. By default, adaption will be performed in all cell zones.

29.12.2. Limiting Adaption by Cell Volume or Volume Weight

The minimum cell volume limit restricts the refinement process to cells with volumes greater than the limit. Use this to initiate the refinement process on larger cells, gradually reducing the limit to create a uniform cell size distribution. Set this limit in the **Min Cell Volume** field. The input that you will give in this field for a 2D axisymmetric problem will be interpreted as the minimum cell area.

In addition, the gradient volume weight can be modified. A value of zero eliminates volume weighting, a value of unity uses the entire volume, and values between 0 and 1 scale the volume weighting. Set this value in the **Volume Weight** field. For more information, see Gradient Adaption Approach in the Theory Guide.

29.12.3. Limiting the Total Number of Cells

The maximum number of cells is a restriction that prevents ANSYS Fluent from creating more cells than required for the present analysis. In addition, it saves the time you would spend waiting for the mesh adaption process to complete the creation of these cells. However, this premature termination of the refinement process can produce undesirable mesh quality depending on the order in which the cells were visited, which is based on the cell arrangement in memory (random).

During the dynamic gradient adaption, the resulting number of cells after adaption is estimated. If this number exceeds the maximum number of cells, both the **Coarsen Threshold** and the **Refine Threshold** are updated. This is done to ensure the best possible mesh resolution with the specified number of cells. You can also specify the minimum number of cells. This is helpful if strong structures of the flow that were resolved with the adaption vanished (for example, left the domain) and you want to resolve the remaining weaker ones. This would otherwise require modifying the **Coarsen Threshold** and the **Refine Threshold**.

You can set the total number of cells allowed in the mesh in the **Max # of Cells** field. The minimum number of cells in the mesh can be set in the **Min # of Cells** field. The default values of zero places no limits on the number of cells.

Note

When using the parallel solver, the **Min # of Cells** and **Max # of Cells** values are not strictly obeyed, but provide an approximate limit to the minimum and maximum cell counts that the adaption algorithm will allow.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

29.12.4. Controlling the Levels of Refinement During Hanging Node Adaption

You can control the number of levels of refinement used to split cells during nonconformal adaption by setting the **Max Level of Refine**. The default value of 2 is a good start for most problems. If this is not sufficient, you can increase this value.

Note

Even in a 2D problem, the default value of 2 can increase the number of cells by a factor of 16 in the adapted regions.

A value of zero leaves this parameter unbounded, and you should use a suitable limit for **Min Cell Volume**. For more information on hanging node adaption, see Hanging Node Adaption in the Theory Guide. For guidelines for limiting cell sizes and number of cells during dynamic gradient adaption, see Dynamic Gradient Adaption Approach (p. 1690).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 30: Creating Surfaces and Cell Registers for Displaying and Reporting Data

ANSYS Fluent enables you to select portions of the domain to be used for visualizing the flow field. The domain portions are called *surfaces*, and there are many ways to create them. Surfaces are required for graphical analysis of 3D problems because you cannot display vectors, contours, and so on, or create an XY plot for the entire domain at once. In 2D you can usually visualize the flow field on the entire domain, but to create an XY plot of a variable in a portion of the interior of the domain, you must generate a surface. In addition, in both 2D and 3D, you will need one or more surfaces if you want to generate a surface-integral report. Note that ANSYS Fluent will automatically create a surface for each boundary zone in the domain. Surface information is stored in the case file.

While you can create surfaces to visualize portions of the domain, you can create cell registers to visualize and perform operations on cell groupings. Similar to adaption registers (see Adapting the Mesh (p. 1681)), you can group cells by:

- Region
- Boundary
- Variable Limiter
- Field Variable
- Residuals
- Volume
- Yplus/Ystar

Cell registers are saved with the case file and can be used to display marked cells, adapt the mesh, apply poor mesh numerics, and report on the cells.

The following sections explain creating surfaces and cell registers:

30.1. Using Surfaces
30.2. Using Cell Registers

30.1. Using Surfaces

In order to visualize the internal flow of a 3D problem or create XY plots of solution variables for 3D results, you must select portions of the domain (surfaces) on which the data is to be displayed. Surfaces can also be used for visualizing or plotting data for 2D problems, and for generating surface-integral reports.

ANSYS Fluent provides methods for creating several kinds of surfaces, and stores all surfaces in the case file. These surfaces and their uses are described briefly below:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Zone Surfaces:

If you want to create a surface that will contain the same cells/faces as an existing cell/face zone, you can generate a zone surface. This kind of surface is useful for displaying results on boundaries.

Partition Surfaces:

When you are using the parallel version of ANSYS Fluent, you may find it useful to create surfaces that are defined by the boundaries between mesh partitions. You can then display data on each side of a partition boundary.

See Parallel Processing (p. 2009) for more information about running the parallel solver.

Point Surfaces:

To monitor the value of some variable or function at a particular location in the domain, you can create a surface consisting of a single point.

Line and Rake Surfaces:

To generate and display pathlines, you must specify a surface from which the particles are released. Line and rake surfaces are well-suited for this purpose and for obtaining data for comparison with wind tunnel data. A rake surface consists of a specified number of points equally spaced between two specified endpoints. A line surface is simply a line that includes the specified endpoints and extends through the domain; data points will be at the centers of the cells through which the line passes, and consequently will not be equally spaced.

Plane Surfaces:

If you want to display flow-field data on a specific plane in the domain, you can create a plane surface. A plane surface is simply a plane that passes through three specified points.

Quadric Surfaces:

To display data on a line (2D), plane (3D), circle (2D), sphere (3D), or quadric surface you can specify the surface by entering the coefficients of the quadric function that defines it. This feature provides you with an explicit method for defining surfaces.

Isosurfaces:

You can use an isosurface to display results on cells that have a constant value for a specified variable. Generating an isosurface based on x, y, or z coordinate, for example, will give you an x, y, or z cross-section of your domain. Generating an isosurface based on pressure will enable you to display data for another variable on a surface of constant pressure.

30.1.1. Zone Surfaces

Zone surfaces are useful for displaying results on boundaries. For example, you may want to plot contours of velocity magnitude at the inlet and outlet of the problem domain, or temperature contours on the domain's walls. To do so, you need to have a surface that contains the same faces (or cells) as an existing face (or cell) zone. Zone surfaces are created automatically for all boundary face zones in the domain, so you will generally not need to create any zone surfaces unless you accidentally delete one.

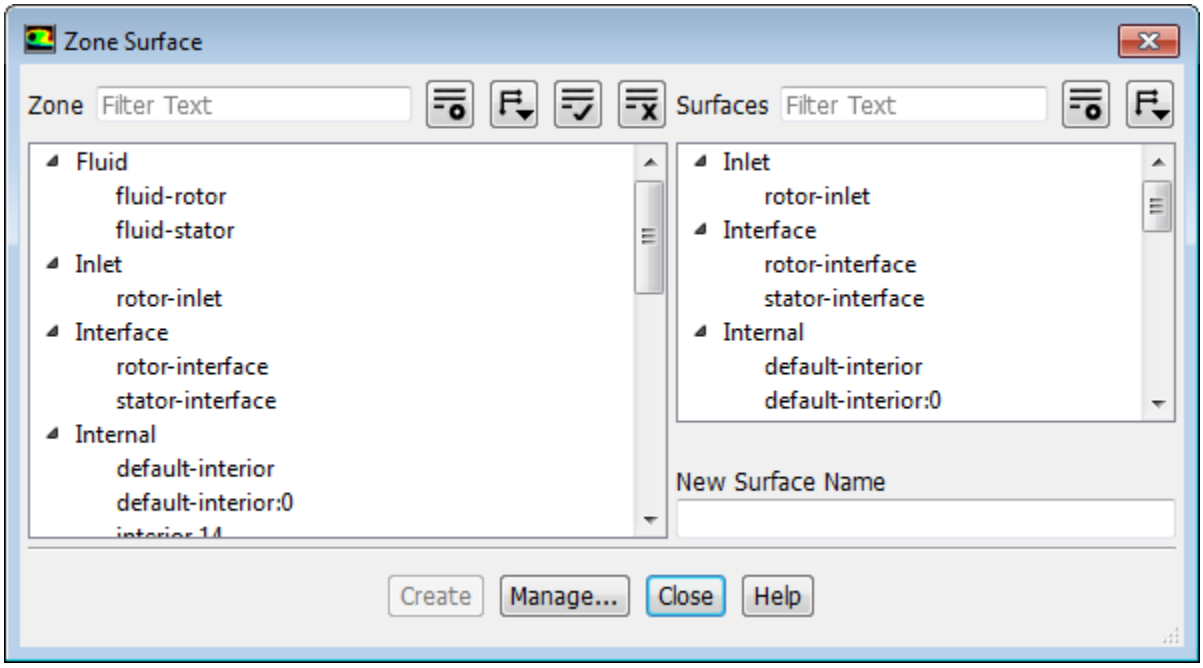
Use the text command: `surface/reset-zone-surfaces` to recreate any missing surface zones by resetting the case surface list.

To create a zone surface, you will use the Zone Surface Dialog Box (p. 2844) (Figure 30.1: The Zone Surface Dialog Box (p. 1711)).

Setting Up Domain → Surface → Create → Zone...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 30.1: The Zone Surface Dialog Box



The steps for creating the zone surface are as follows:

1. Select the zone(s) in the **Zone** list.

2. Enter a name in **New Surface Name** if you do not want ANSYS Fluent to assign a default name to the newly created surface. By default, Fluent assigns each surface the same name as the zone that it corresponds to. If a surface already exists with that name, Fluent assigns the surface a unique name using the `zone-surface` prefix (for example, `zone-surface-6`).

   If you do not want to use the default name, enter a new name in **New Surface Name**. If the **New Surface Name** that you enter is the same as a surface that already exists, Fluent assigns the new surface a unique name using the `zone-surface` prefix (for example, `zone-surface-7`).

   Important

   - The surface name that you enter must begin with an alphabetical letter. If your surface name begins with any other character or number, Fluent rejects the entry.
   - If you are selecting multiple zones, you do not have the option to choose the surface names. ANSYS Fluent automatically give the surfaces the same names as their corresponding zone.

3. Click **Create**. The new surface name is added to the **Surfaces** list in the dialog box.

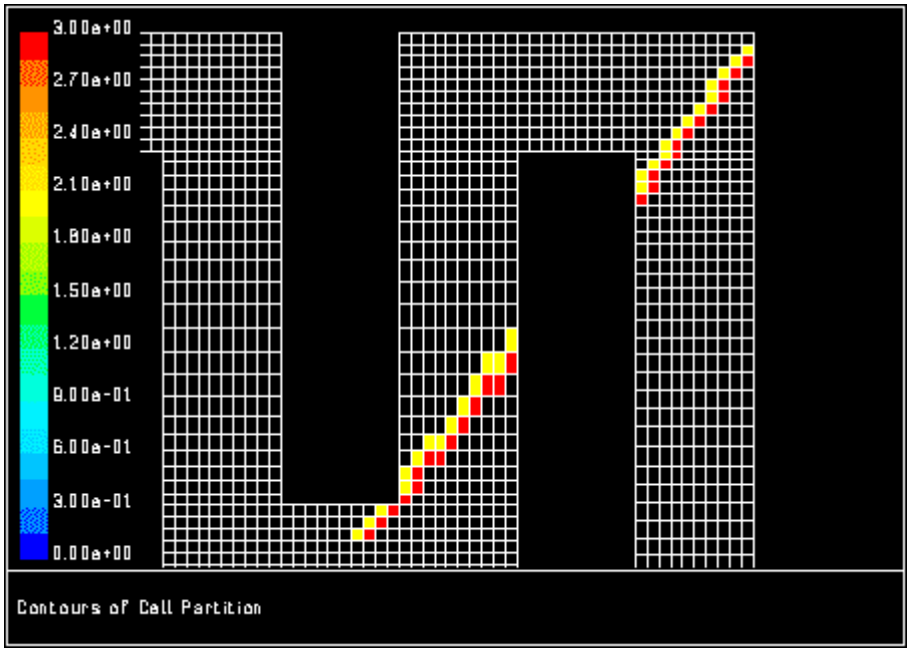
If you want to delete or otherwise manipulate any surfaces, click **Manage...** to open the Surfaces Dialog Box (p. 2810). For details, see Grouping, Renaming, and Deleting Surfaces (p. 1734).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

30.1.2. Partition Surfaces

If you are using the parallel version of ANSYS Fluent (see Parallel Processing (p. 2009)), you may find it useful to create data surfaces defined by the boundaries of mesh partitions. As described in Mesh Partitioning and Load Balancing (p. 2030), partitioning the mesh divides it into groups of cells that can be solved on separate processors when you use a parallel solver. A partition surface will contain faces or cells on the boundary of two mesh partitions. For example, you can plot solution values on the partition surface to determine how the solution is changing across a partition interface, as shown in the following figure:

Figure 30.2: Contours of Cell Partitions on Partition Surface Overlaid on Mesh

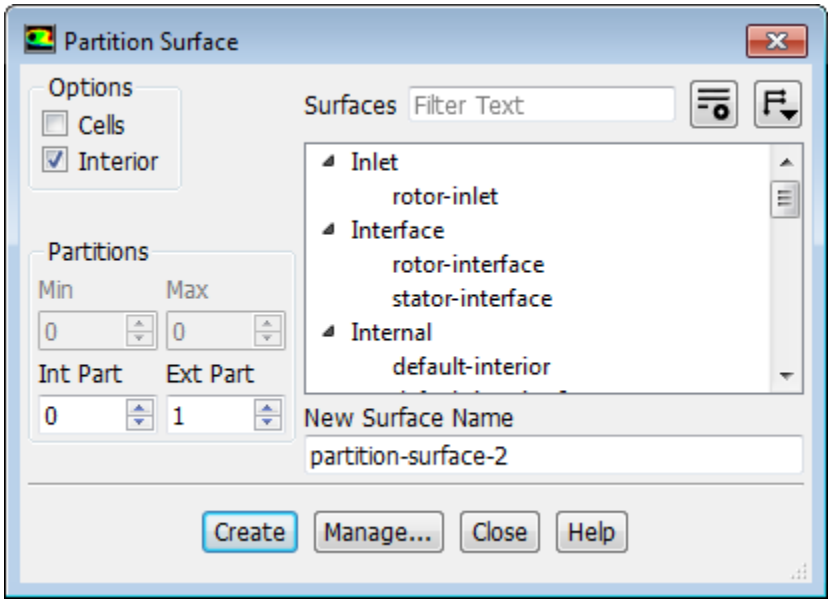


To create a partition surface, you will use the Partition Surface Dialog Box (p. 2766) (Figure 30.3: The Partition Surface Dialog Box (p. 1713)).

Setting Up Domain → Surface → Create → Partition...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 30.3: The Partition Surface Dialog Box



The procedure for creating the partition surface are as follows:

1. Specify the partition boundary in which you are interested by indicating the two bordering partitions under the **Partitions** heading. The boundary that defines the partition surface is the boundary between the "interior partition" and the "exterior partition". **Int Part** indicates the ID number of the interior partition (that is, the partition under consideration), and **Ext Part** indicates the ID number of the bordering (exterior) partition. The **Min** and **Max** fields will indicate the minimum and maximum ID numbers of the mesh partitions. The minimum is always zero, and the maximum is one less than the number of processors. If there are more than two mesh partitions, each interior partition will share boundaries with several exterior partitions. By setting the appropriate values for **Int Part** and **Ext Part**, you can create surfaces for any of these boundaries.

2. Choose interior or exterior faces or cells to be contained in the partition surface by selecting or clearing **Cells** and **Interior** under **Options**. To obtain a surface consisting of cells that are on the "interior" side of the partition boundary, select both **Cells** and **Interior**. To create one consisting of cells that are on the "exterior" side, select **Cells** and clear **Interior**. If you want the surface to contain the faces on the boundary instead of the cells, clear the **Cells** option. To have the faces reflect data values for the interior cells, select the **Interior** check box, and to have them reflect values for the exterior cells, clear it.

3. If you do not want to use the default name assigned to the surface, enter a new name under **New Surface Name**. The default name is the concatenation of the surface type and an integer that is the new surface ID (for example, `partition-surface-6`). (If the **New Surface Name** you enter is the same as the name of a surface that already exists, ANSYS Fluent will automatically assign the default name to the new surface when it is created.)

   Important

   The surface name that you enter must begin with an alphabetical letter. If the name of the surface begins with any other character or number, ANSYS Fluent rejects the entry.

4. Click **Create**. The new surface name is added to the **Surfaces** list in the dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you want to delete or otherwise manipulate any surfaces, click **Manage...** to open the Surfaces Dialog Box (p. 2810). For details, see Grouping, Renaming, and Deleting Surfaces (p. 1734).

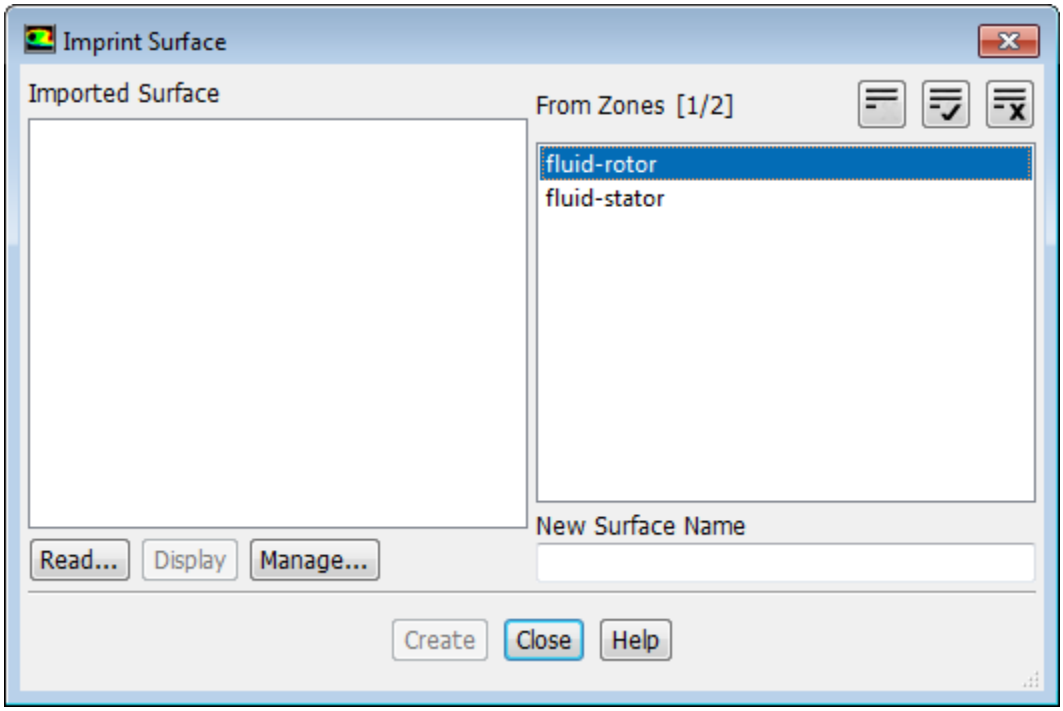
30.1.3. Imprint Surfaces

If you want to display results on surfaces that are more complicated than a simple point or plane, then you can import a custom surface representing the region of interest (`.stl` or `.msh` format) and "imprint" this surface into the current mesh. For example, you may want to visualize only a smaller custom portion of a much larger (complex) geometry.

To create an imprinted surface, use the Imprint Surface Dialog Box (p. 2714) (Figure 30.4: The Imprint Surface Dialog Box (p. 1714)).

Setting Up Domain → Surface → Create → Imprint...

Figure 30.4: The Imprint Surface Dialog Box



Create an imprinted surface as follows:

1. Click **Read** and select your geometry file (`.stl` or `.msh` format) using the **Select File** dialog box.

   Specify the units the mesh was created in (default is meters).

2. Click **OK** to upload your file.

   The name of your geometry file appears in the **Imported Surface** *dialog box.*

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select the zone where your geometry is located in the **From Zones** group box.

> **Note**
>
> If you do not specify a zone, Fluent uses the entire domain for imprinting. For a large mesh this could be time consuming.

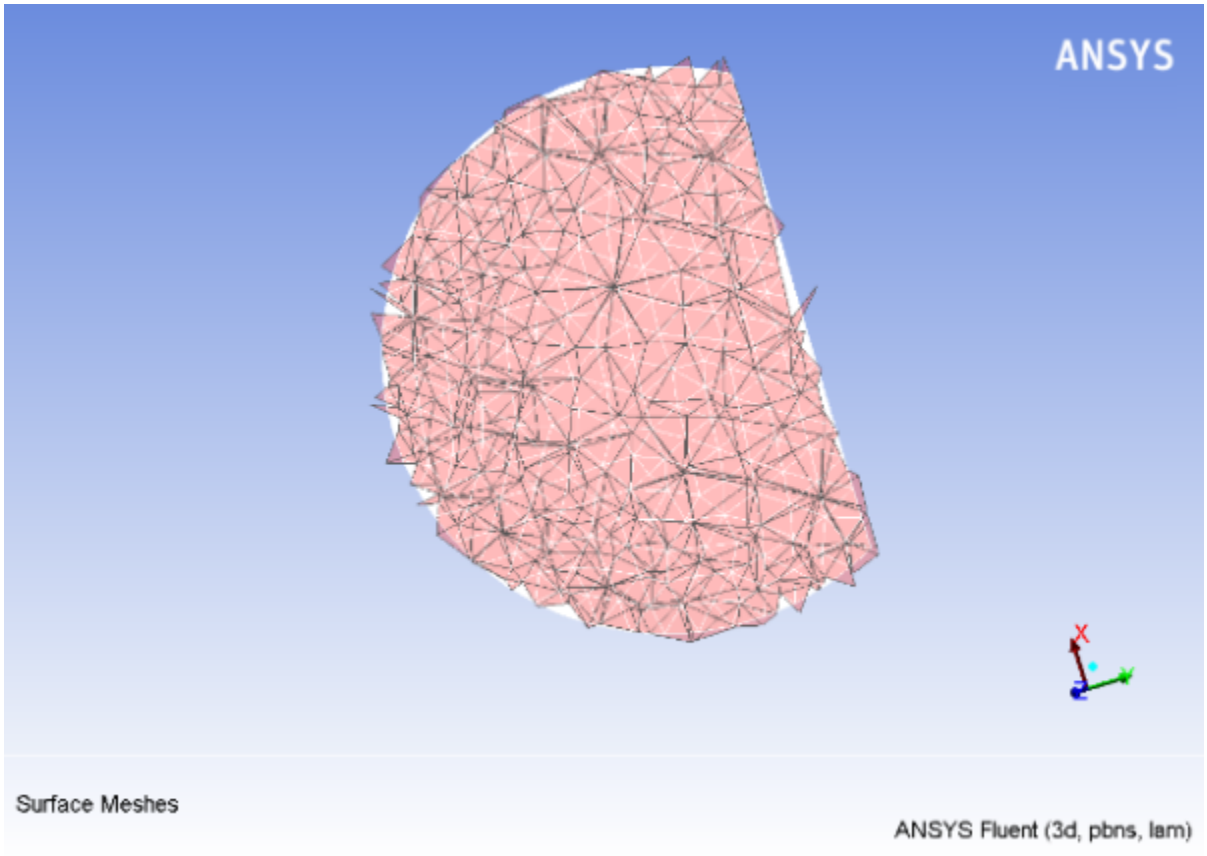
4. Enter a name for your new surface in the **New Surface Name** dialog box.

   The default name is `<orig-name>(imprint)<id>`.

5. Click **Create** to create the new surface.

When an imported surface is imprinted, a postprocessing surface that coincides with the imported surface is created. The imprinted surface will not have facets for the portions of the original surface that lie outside the domain. Different postprocessing operations such as contour and vector plots, and surface integrals can be performed on this surface. However, the edges of the imprinted surface may not coincide with those of the original surface, if the edges of the original surface do not coincide with mesh edges. An example is shown in the following picture.

Figure 30.5: Imprinted Surface (pink) Superimposed Over Imported Surface (white)



As you can see in Figure 30.5: Imprinted Surface (pink) Superimposed Over Imported Surface (white) (p. 1715), there are deformities around the edges of the imprinted surface.

Important

If you want to create an imprint surface from multiple disconnected source surfaces, it is better to create multiple imprint surfaces (one per source surface) and, if required, group them together after they are created. For information on grouping surfaces, see Grouping, Renaming, and Deleting Surfaces (p. 1734).

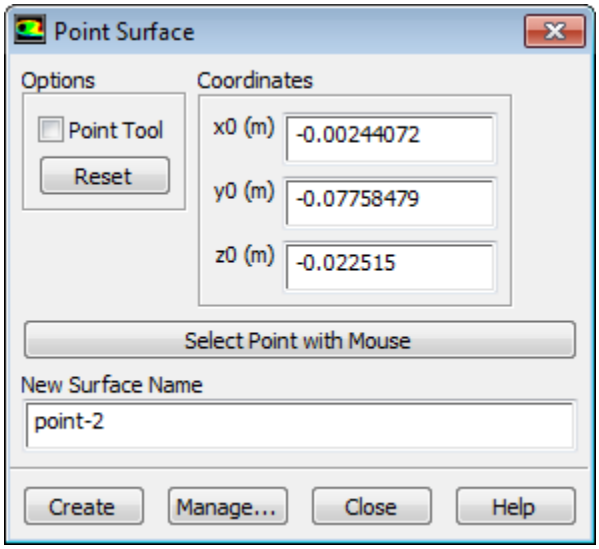
30.1.4. Point Surfaces

You may often be interested in displaying results at a single point in the domain. For example, you may want to monitor the value of some variable or function at a particular location. To do this, you must first create a "point" surface, which consists of a single point. When you display node-value data on a point surface, the value displayed is a linear average of the neighboring node values. If you display cell-value data, the value at the cell in which the point lies is displayed.

To create a point surface, use the Point Surface Dialog Box (p. 2780) (Figure 30.6: The Point Surface Dialog Box (p. 1716)).

Setting Up Domain → Surface → Create → Point...

Figure 30.6: The Point Surface Dialog Box



Create a point surface as follows:

1. Specify the location of the point. There are three different ways to do this:
   - Enter the coordinates (**x0, y0, z0**) under **Coordinates**.
   - Click **Select Point With Mouse** and then select the point by clicking on a location in the active graphics window with the mouse-probe button. (See Controlling the Mouse Button Functions (p. 1800) for information about setting mouse button functions.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Use the **Point Tool** option to interactively position a point in the graphics window. You can set the initial location of this point using one of the two methods described above for specifying the point's position (or you can start from the position defined by the default **Coordinates**). See Using the Point Tool (p. 1717) for information about using the point tool.

2. If you do not want to use the default name assigned to the surface, enter a new name under **New Surface Name**. The default name is the concatenation of the surface type and an integer that is the new surface ID (for example, `point-5`). (If the **New Surface Name** you enter is the same as the name of a surface that already exists, ANSYS Fluent will automatically assign the default name to the new surface when it is created.)

   Important

   The surface name that you enter must begin with an alphabetical letter. If the name of the surface begins with any other character or number, ANSYS Fluent rejects the entry.

3. Click **Create** to create the new surface.

If you want to check that your new surface has been added to the list of all defined surfaces, or you want to delete or otherwise manipulate any surfaces, click **Manage...** to open the Surfaces Dialog Box (p. 2810). For details, see Grouping, Renaming, and Deleting Surfaces (p. 1734) for details.

30.1.4.1. Using the Point Tool

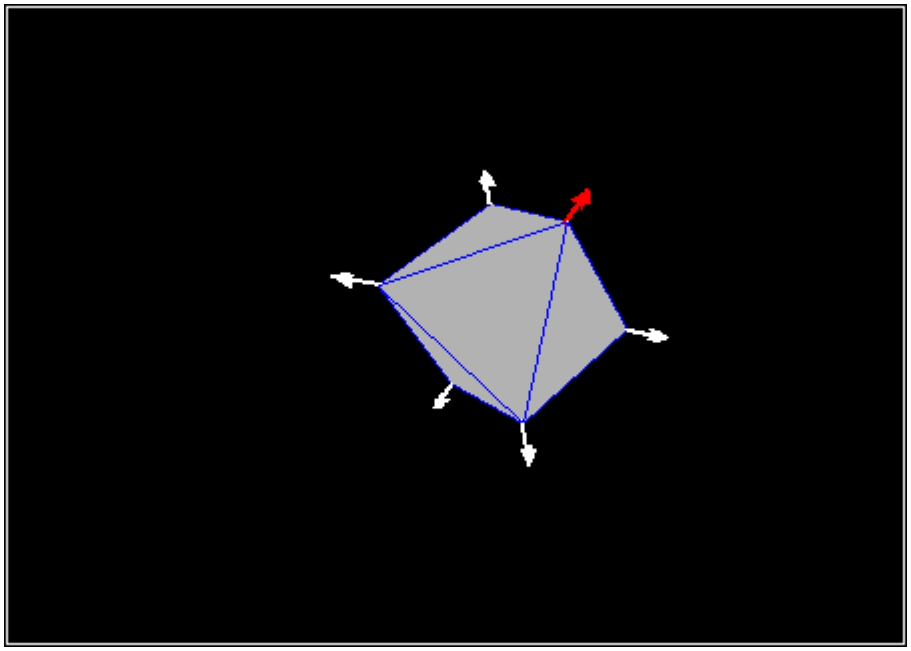
The point tool enables you to interactively fine-tune the definition of a point using graphics. Starting from an initial point, you can translate the point until its position is as desired. For example, if you need to position a point surface at the center of a duct, just past the inlet, you can start with the point tool near the desired location (such as on the inlet), and translate it until it is in the proper place. (You may find it helpful to display mesh faces to ensure that the point tool is correctly positioned inside the domain.)

30.1.4.1.1. Initializing the Point Tool

Before enabling the **Point Tool** option, set the **Coordinates** to suitable starting values. You can enter values manually, or use the **Select Point With Mouse** button. Often it is convenient to display the mesh for an inlet or isosurface on or near where the point is to be located, and then select a point on that mesh to specify the initial position of the point tool. Once you have specified the appropriate **Coordinates**, turn on the tool by enabling the **Point Tool** option. The point tool, an eight-sided polygon, will appear in the graphics window, as shown in Figure 30.7: The Point Tool (p. 1718).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 30.7: The Point Tool



You can then translate the point tool as described below. The point surface you create will be located at the center of the point tool.

30.1.4.1.2. Translating the Point Tool

To translate the point tool in the direction along the red axis, click the mouse-probe button (the right button by default) anywhere on the gray part of the point tool and drag the mouse until the tool reaches the desired location. Green arrows will show the direction of motion. For information about changing the mouse functions, see Controlling the Mouse Button Functions (p. 1800).

To translate the tool in the transverse directions (that is, along either of the other axes), press **Shift**, click the mouse-probe button anywhere on the gray part of the point tool, and drag the mouse until the tool reaches the desired location. Two sets of green arrows will show the possible directions of motion. In 2D, there will be only one set of green arrows because there is only one other direction for translation. If you find the perspective distracting when performing this type of translation, you can turn it off in the Camera Parameters Dialog Box (p. 2587) (opened from the Views Dialog Box (p. 2583)), as described in Controlling Perspective and Camera Parameters (p. 1808).

30.1.4.1.3. Resetting the Point Tool

If you "lose" the point tool, or want to reset it for any other reason, you can either click **Reset** to return the point tool to the default position and start from there, or turn the tool off and re-initialize it as described above. In the default position, the point tool will lie at the center of the domain.

30.1.5. Line and Rake Surfaces

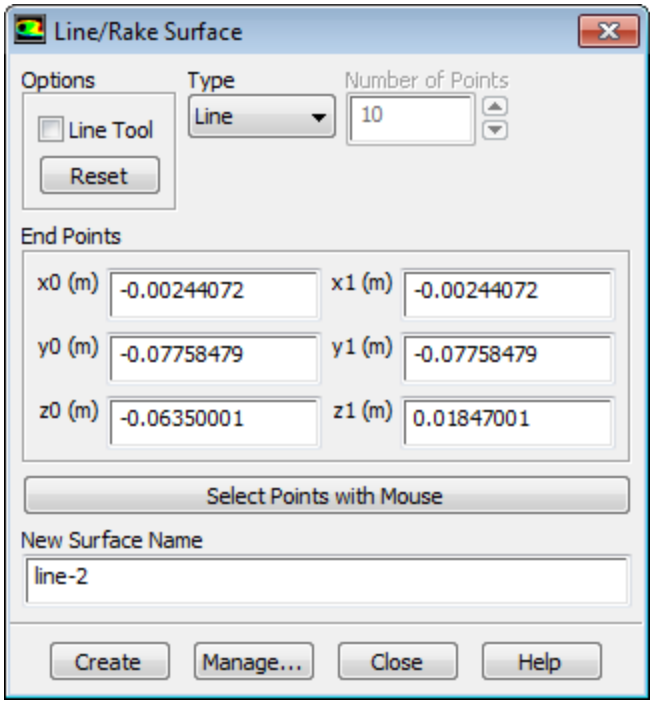
You can create lines and rakes in the domain for releasing particles, obtaining data for comparison with tunnel data, and so on. A rake consists of a specified number of points equally spaced between two specified endpoints. A line is simply a line that extends up to and includes the specified endpoints; data points will be located where the line intersects the faces of the cell, and consequently may not be equally spaced.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To create a line or rake surface, you will use the Line/Rake Surface Dialog Box (p. 2727) (Figure 30.8: The Line/Rake Surface Dialog Box (p. 1719)).

Setting Up Domain → Surface → Create → Line/Rake...

Figure 30.8: The Line/Rake Surface Dialog Box



The steps for creating the line or rake surface are as follows:

1. Indicate whether you are creating a **Line** surface or a **Rake** surface by selecting the appropriate item in the **Type** drop-down list.

2. If you are creating a rake surface, specify the **Number of Points** to be equally spaced between the two endpoints.

3. Specify the location of the line or rake surface. There are three different ways to define the location:

   - Enter the coordinates of the first point (**x0, y0, z0**) and the last point (**x1, y1, z1**) under **End Points**.

   - Click **Select Points With Mouse** and then select the endpoints by clicking on locations in the active graphics window with the mouse-probe button. (See Controlling the Mouse Button Functions (p. 1800) for information about setting mouse button functions.)

   - Use the **Line Tool** option to interactively position a line in the graphics window. You can set the initial location of this line using one of the two methods described above for specifying endpoints (or you can start from the position defined by the default **End Points**). See Using the Line Tool (p. 1720) for information about using the line tool.

   Note that when you use the second or third method described above, the coordinates of the **End Points** are updated automatically.

4. If you do not want to use the default name assigned to the surface, enter a new name under **New Surface Name**. The default name is the concatenation of the surface type and an integer that is the new surface

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ID (for example, `line-5` or `rake-6`). If the **New Surface Name** you enter is the same as the name of a surface that already exists, ANSYS Fluent will automatically assign the default name to the new surface when it is created.

Important

The surface name that you enter must begin with an alphabetical letter. If the name of the surface begins with any other character or number, ANSYS Fluent rejects the entry.

5. Click **Create** to create the new surface.

If you want to check that the new surface has been added to the list of all defined surfaces, or you want to delete or otherwise manipulate any surfaces, click **Manage...** to open the Surfaces Dialog Box (p. 2810). See Grouping, Renaming, and Deleting Surfaces (p. 1734) for details.

30.1.5.1. Using the Line Tool

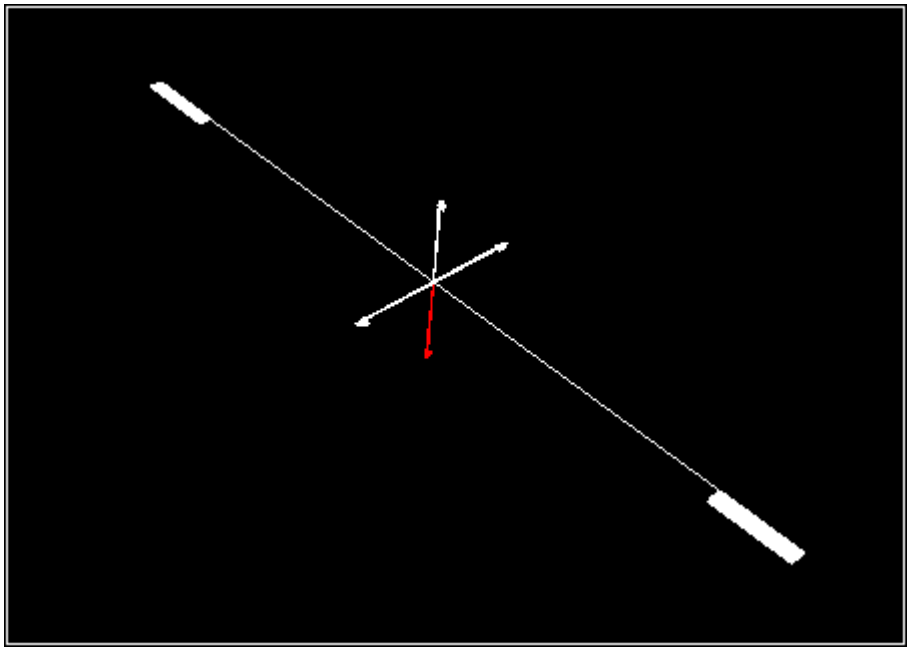
The line tool enables you to interactively fine-tune the definition of a line or rake using graphics. Starting from an initial line, you can translate, rotate, and resize the line until its position, orientation, and length are as desired. For example, if you need to position a rake surface just inside the inlet to a duct, you can start with the line tool near the desired location (such as on the inlet), and translate, rotate, and resize it until you are satisfied. You may find it helpful to display mesh faces to ensure that the line tool is correctly positioned inside the domain.

30.1.5.1.1. Initializing the Line Tool

Before enabling the **Line Tool** option, set the **End Points** to suitable starting values. You can enter values manually, or use the **Select Points With Mouse** button. Often it is convenient to display the mesh for an inlet or isosurface on or near where you want to place the line or rake surface and then select two points on that mesh to specify the initial position of the line tool. Once you have specified the appropriate **End Points**, enable the **Line Tool** option. The line tool appears in the graphics window, as shown in Figure 30.9: The Line Tool (p. 1721).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 30.9: The Line Tool



You can then translate, rotate, and/or resize the line tool as described in the following sections.

30.1.5.1.2. Translating the Line Tool

To translate the line tool in the direction along the red axis, click the mouse-probe button (the right button by default) anywhere on the "line" part of the tool and drag the mouse until the tool reaches the desired location. Green arrows will show the direction of motion For information about changing the mouse functions see Controlling the Mouse Button Functions (p. 1800).

Important

Do not click the axes of the line tool that have arrows on the ends. These axes control rotation of the tool. Click only on the portion of the tool that represents the prospective line surface. This portion is designated by the rectangles attached to each end.

To translate the tool in the transverse directions (that is, along either of the axes within the plane perpendicular to the red axis), press **Shift**, click the mouse-probe button anywhere on the "line" part of the tool, and drag the mouse until the tool reaches the desired location. Two sets of green arrows will show the possible directions of motion. In 2D, there will be only one set of green arrows because there is only one other direction for translation. If you find the perspective distracting when performing this type of translation, you can disable it in the Camera Parameters Dialog Box (p. 2587) (opened from the Views Dialog Box (p. 2583)), as described in Controlling Perspective and Camera Parameters (p. 1808).

30.1.5.1.3. Rotating the Line Tool

To rotate the line tool, you click the mouse-probe button on one of the white axes with arrows. When you click one of these axes, a green ribbon will encircle the other arrowed axis, designating it as the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

axis of rotation. As you drag the mouse along the circle to rotate the tool, the green circle will become yellow.

Important

Do not click the red axis to rotate the line tool.

30.1.5.1.4. Resizing the Line Tool

If you plan to generate a rake surface, you can resize the line tool to define the length of the rake. Click the mouse-probe button in one of the white rectangles at the ends of the "line" part of the tool (shown in black in Figure 30.9: The Line Tool (p. 1721)) and drag the mouse to lengthen or shorten the tool. Green arrows will show the direction of stretching/shrinking.

30.1.5.1.5. Resetting the Line Tool

If you "lose" the line tool, or want to reset it for any other reason, you can either click **Reset** to return the line tool to the default position and start from there, or turn the tool off and re-initialize it as described above. In the default position, the line tool will lie midway along the x and y lengths of the domain, spanning the z domain extent.

30.1.6. Plane Surfaces

To display flow-field data on a specific plane in the domain, you will use a plane surface. You can create surfaces that cut through the solution domain along arbitrary planes only in 3D; this feature is not available in 2D.

There are six types of plane surfaces that you can create:

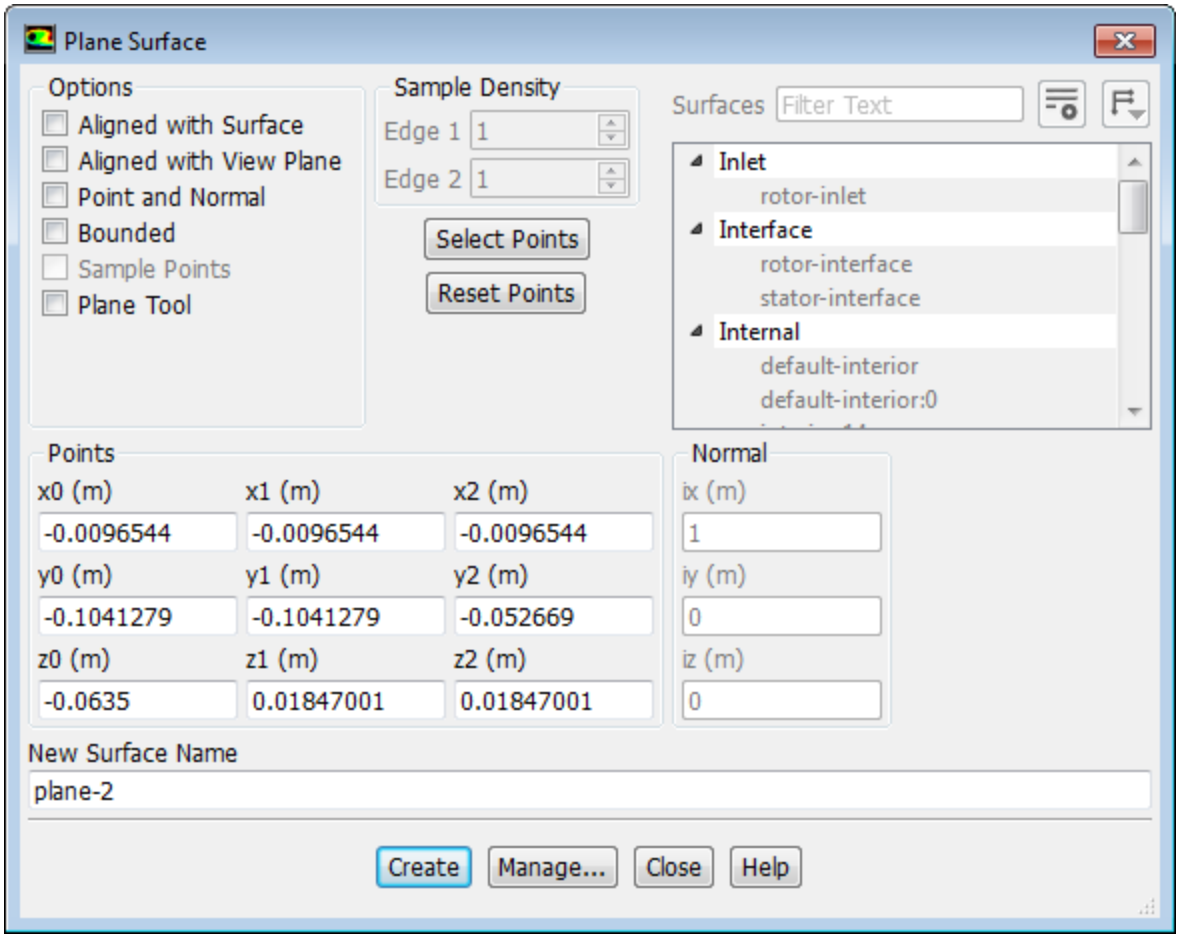
- Intersection of the domain with the infinite plane: This is the default plane surface created. The extents of the plane is determined by the extents of the domain. Because the plane is slicing through the domain, the data points will, by default, be located where the plane intersects the faces of a cell, and consequently may not be equally spaced.
- Bounded plane: This plane will be a bounded parallelepiped, for which 3 of the 4 corners are the 3 points that define the plane equation (or the 4 corners are the corners of the "plane tool"). Like the default plane surface described above, this type of surface will also have unequally spaced data points.
- Bounded plane with equally spaced data points: This plane is the same as the bounded plane described above, except you will specify the density of points along the two directions of the parallelepiped, creating a uniform distribution of data points.
- Plane having a certain normal vector and passing through a specified point: To create this type of plane, define a normal vector and a point. A plane with the specified normal and passing through the specified point will be created.
- Plane aligned with an existing surface: To create this type of plane, you will define a single point and a surface. A plane parallel to the selected surface and passing through the specified point will be created.
- Plane aligned with the view in the graphics window: To create this type of plane, you will define a single point. A plane parallel to the current view in the active graphics window and passing through the specified point will be created.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To create a plane surface, you will use the Plane Surface Dialog Box (p. 2777) (Figure 30.10: The Plane Surface Dialog Box (p. 1723)).

Setting Up Domain → Surface → Create → Plane...

Figure 30.10: The Plane Surface Dialog Box



The procedure for creating the plane surface is as follows:

1. Decide which of the six types of planes you want to create.

   - To create the default plane type (the intersection of the infinite plane with the domain), go directly to step 2.
   - To create a bounded plane, select **Bounded** under **Options**.
   - To create a bounded plane with equally spaced data points, select both **Bounded** and **Sample Points**, and then set the number of data points under **Sample Density**. You will specify the point density in each direction by entering the appropriate values for **Edge 1** and **Edge 2**. Edge 1 extends from point 0 to point 1, and edge 2 extends from point 1 to point 2.
   - To define a plane aligned with an existing surface, select **Aligned With Surface**, and then choose the surface in the **Surfaces** list and specify a single point using one of the first two methods described below in step 2.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To define a plane aligned with the view plane, select **Aligned With View Plane**, and then choose a single point using one of the first two methods described below in step 2.
- To define a plane having a certain normal vector and passing through a specified point, select **Point And Normal**, and then specify the normal vector by entering values in the **ix**, **iy**, and **iz** fields under **Normal**, and a single point using one of the first two methods described below in step 2.

2. Specify the location of the plane surface. There are three different ways to define the location:
   - Enter the coordinates of the three **Points** defining the planar surface: (**x0, y0, z0**), (**x1, y1, z1**), and (**x2, y2, z2**).
   - Click **Select Points** and then select the three points by clicking on locations in the active graphics window with the mouse-probe button. (See Controlling the Mouse Button Functions (p. 1800) for information about setting mouse button functions.)
   - Use the **Plane Tool** option to interactively position a plane in the graphics window. You can set the initial location of this plane using one of the two methods described above for specifying the defining points. You can also start from the position defined by the default **Points**. See Using the Plane Tool (p. 1724) for information about using the plane tool.

   Note that when you use the second or third method described above, the coordinates of the **End Points** are updated automatically.

3. If you do not want to use the default name assigned to the surface, enter a new name under **New Surface Name**. The default name is the concatenation of the surface type and an integer that is the new surface ID (for example, `plane-7`). (If the **New Surface Name** you enter is the same as the name of a surface that already exists, ANSYS Fluent will automatically assign the default name to the new surface when it is created.)

   Important

   The surface name that you enter must begin with an alphabetical letter. If the name of the surface begins with any other character or number, ANSYS Fluent rejects the entry.

4. Click **Create** to create the new surface.

If you want to check that your new surface has been added to the list of all defined surfaces, or you want to delete or otherwise manipulate any surfaces, click **Manage...** to open the Surfaces Dialog Box (p. 2810). See Grouping, Renaming, and Deleting Surfaces (p. 1734) for details.

30.1.6.1. Using the Plane Tool

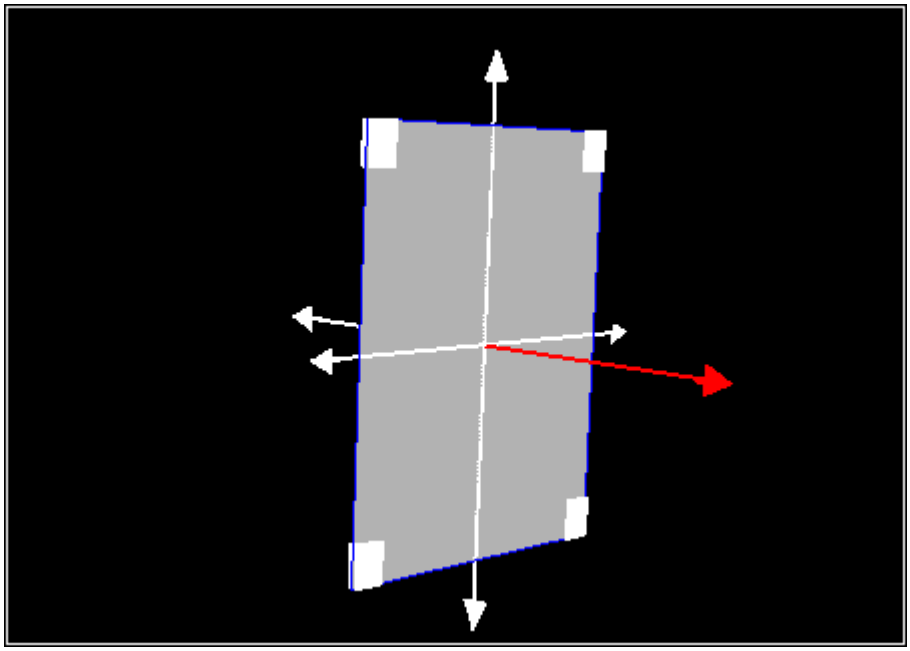
The plane tool enables you to interactively fine-tune the definition of a plane using graphics. Starting from an initial plane, you can translate, rotate, and resize the plane until its position, orientation, and size are as desired. For example, if you need to position a plane surface at a cross-section of an irregularly-shaped, curved duct, you can start with the plane tool near the desired location, resize it, translate it until it is within the duct walls, and rotate it to the proper orientation. You may find it helpful to display mesh faces to ensure that the plane tool is correctly positioned inside the domain.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

30.1.6.1.1. Initializing the Plane Tool

Before enabling the **Plane Tool** option, set the **Points** to suitable starting values. You can enter values manually, or use the **Select Points** button. Often it is convenient to display the mesh for an inlet or isosurface that is similar to the desired plane surface, and then select three points on that mesh to position the initial plane. Once you have specified the appropriate **Points**, enable the **Plane Tool** option. The plane tool will appear in the graphics window, as shown in Figure 30.11: The Plane Tool (p. 1725).

Figure 30.11: The Plane Tool



You can then translate, rotate, and/or resize the plane tool as described in the following sections.

30.1.6.1.2. Translating the Plane Tool

To translate the plane tool in the direction normal to the plane, click the mouse-probe button (the right button by default) anywhere on the gray part of the plane tool and drag the mouse until the tool reaches the desired location. Green arrows will show the direction of motion. For information about changing the mouse functions see Controlling the Mouse Button Functions (p. 1800).

To translate the tool in the transverse directions (that is, along either of the axes that lie within the plane), press **Shift**, click the mouse-probe button anywhere on the gray part of the plane tool, and drag the mouse until the tool reaches the desired location. Two sets of green arrows will show the possible directions of motion. If you find the perspective distracting when performing this type of translation, you can turn it off in the Camera Parameters Dialog Box (p. 2587) (opened from the Views Dialog Box (p. 2583)), as described in Controlling Perspective and Camera Parameters (p. 1808).

30.1.6.1.3. Rotating the Plane Tool

To rotate the plane tool, click the mouse-probe button on one of the white arrows at the tips of the plane's axes. Clicking on any arrow rotates the tool about either of the other two axes: when you click the arrow, two green ribbons will encircle the plane tool, forming circles about each of the two possible axes of rotation. Drag the mouse along the desired circle to rotate the tool. As you do so, the circle along which the tool is rotating will become yellow.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following notes may help you when you are rotating the plane tool:

- Once you move your mouse along one circle, you cannot change the direction of rotation unless you release the mouse-probe button and try again. Be careful to start moving your mouse very steadily so that you can choose the correct direction.
- Do not click the red arrow to rotate.
- Do not try to rotate by clicking on an arrow that is pointing away from you. It will be very difficult for you to judge which direction of rotation is correct from this point of view. Because there are two arrows on each axis, there will always be an appropriate arrow available.
- Do not rotate the plane tool more than 90° or so at once. If you rotate the tool by a large angle, the arrow on which you are clicking will begin to point away from you, and you will have trouble controlling the rotation (as discussed in the item above).

30.1.6.1.4. Resizing the Plane Tool

If you plan to generate a bounded plane, you can resize the plane tool to define the plane's boundaries. Click the mouse-probe button in one of the white squares at the plane tool's corners (shown in black in Figure 30.11: The Plane Tool (p. 1725)) and drag the mouse to stretch or shrink the tool. Green arrows will show the direction of the plane's diagonal.

Important

Be careful not to drag your mouse across any of the axes while resizing the tool. This will flip the tool over and corrupt it. If you accidentally do this, reset the plane tool and start again.

30.1.6.1.5. Resetting the Plane Tool

If you "lose" the plane tool, or want to reset it for any other reason, you can either click **Reset Points** to return the plane tool to the default position and start from there, or turn the tool off and re-initialize it as described above. In the default position, the plane tool will lie midway along the x length of the domain, spanning the y and z domain extents.

30.1.7. Quadric Surfaces

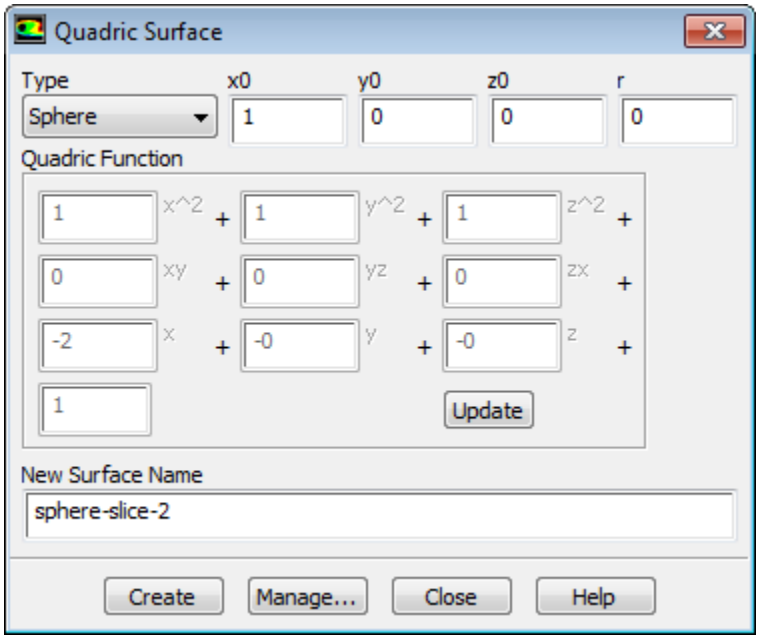
If you want to display data on a line (2D), plane (3D), circle (2D), sphere (3D), or general quadric surface, you can specify the surface by entering the coefficients of the quadric function that defines it. This feature provides you with an explicit method for defining surfaces. See Line and Rake Surfaces (p. 1718) and Plane Surfaces (p. 1722) for additional methods for creating line and plane surfaces.

To create a quadric surface, you will use the Quadric Surface Dialog Box (p. 2781) (Figure 30.12: The Quadric Surface Dialog Box (p. 1727)).

Setting Up Domain → Surface → Create → Quadric...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 30.12: The Quadric Surface Dialog Box



The steps for creating the quadric surface are as follows:

1. Decide which type of quadric surface you want to create. In 3D, choose **Plane**, **Sphere**, or (general) **Quadric** in the **Type** drop-down list. In 2D, choose **Line**, **Circle**, or **Quadric**.

2. Specify the defining equation for the surface *in SI units*.

   - Line or plane surface: If you have selected **Line** (in 2D) or **Plane** (in 3D) as the surface type, the surface will consist of all points on the domain that satisfy the equation $ix*x+iy*y+iz*z=\text{distance}$. You will specify **ix** (the coefficient of x), **iy** (the coefficient of y), **iz** (the coefficient of z), and **distance** (the distance of the line or plane from the origin) in the fields to the right of the **Type** drop-down list. When you click **Update** under the **Quadric Function** heading, the display of the quadric function coefficients will change to reflect your inputs.

   - Circle or sphere surface: If you have selected **Circle** (in 2D) or **Sphere** (in 3D) as the surface type, the surface will consist of all points on the domain that satisfy the equation $(x-x0)^2+(y-y0)^2+(z-z0)^2=r^2$. You will specify **x0**, **y0**, **z0** (the x, y, and z coordinates of the sphere or circle's center) and **r** (the radius) in the fields to the right of the **Type** drop-down list. When you click **Update** under the **Quadric Function** heading, the display of the quadric function coefficients will change to reflect your inputs.

   - Quadric surface: If you have selected **Quadric** as the surface type, the surface will consist of all points in the domain that satisfy the general quadric function $Q=\text{value}$. You will specify the coefficients of the quadric function Q (the coefficients of the terms x_2, y_2, z_2, xy, yz, zx, x, y, z and the constant term) directly in the **Quadric Function** box, and you will set **value** to the right of the **Type** drop-down list. Note that the **Update** button will be disabled when you choose this type of surface.

3. If you do not want to use the default name assigned to the surface, enter a new name under **New Surface Name**. The default name is the concatenation of the surface type and an integer that is the new surface ID (for example, `sphere-slice-7` or `quadric-slice-10`). If the **New Surface Name** you enter is

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the same as the name of a surface that already exists, ANSYS Fluent will automatically assign the default name to the new surface when it is created.

Important

The surface name that you enter must begin with an alphabetical letter. If the name of the surface begins with any other character or number, ANSYS Fluent rejects the entry.

4. Click **Create** to create the new surface.

If you want to check that your new surface has been added to the list of all defined surfaces, or you want to delete or otherwise manipulate any surfaces, click **Manage...** to open the Surfaces Dialog Box (p. 2810). See Grouping, Renaming, and Deleting Surfaces (p. 1734) for details.

30.1.8. Isosurfaces

If you want to display results on cells that have a constant value for a specified variable, you will need to create an isosurface of that variable. Generating an isosurface based on x, y, or z coordinate, for example, will give you an x, y, or z cross-section of your domain; generating an isosurface based on pressure will enable you to display data for another variable on a surface of constant pressure. You can create an isosurface from an existing surface or from the entire domain. Furthermore, you can restrict any isosurface to a specified cell zone.

Important

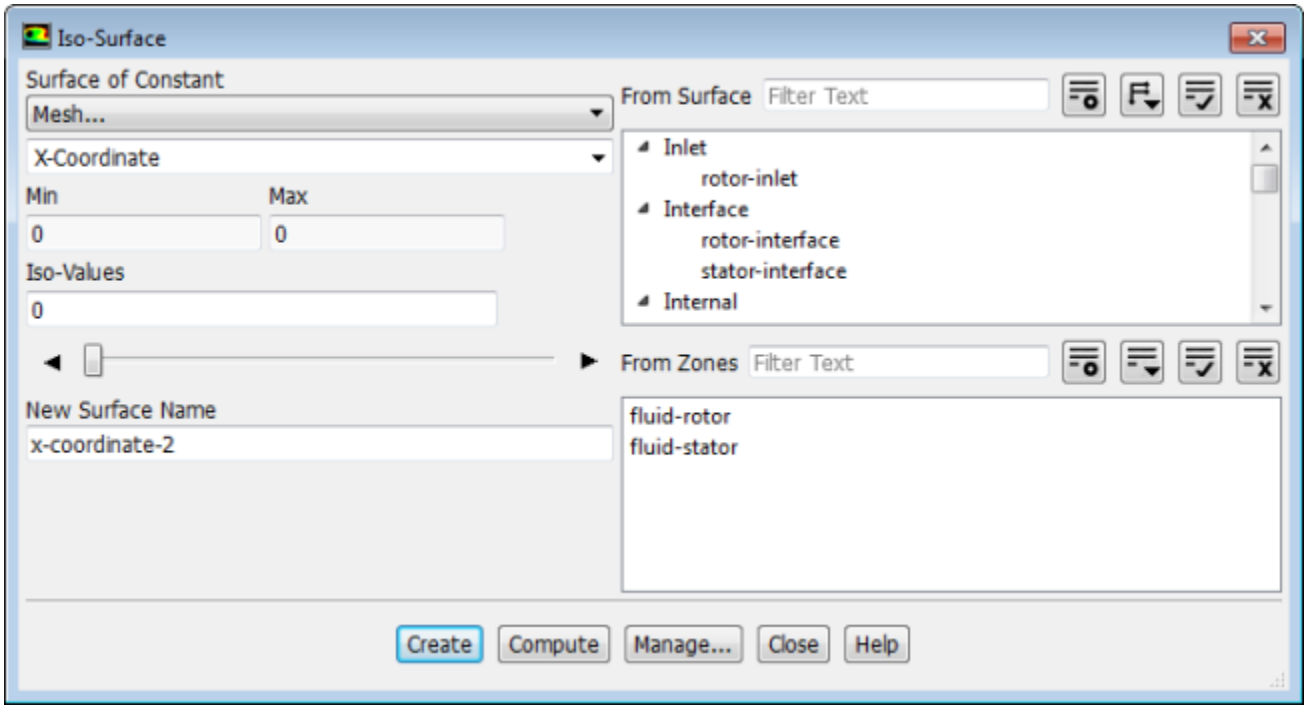
Note that you cannot create an isosurface until you have initialized the solution, performed calculations, or read a data file.

To create an isosurface, you will use the Iso-Surface Dialog Box (p. 2721) (Figure 30.13: The Iso-Surface Dialog Box (p. 1729)).

Setting Up Domain → Surface → Create → Iso-Surface...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 30.13: The Iso-Surface Dialog Box



The steps for creating the isosurface are as follows:

1. Choose the scalar variable to be used for isosurfacing in the **Surface of Constant** drop-down list. First, select the desired category in the upper list. You can then select from related quantities from the lower list. (See Field Function Definitions (p. 1931) for an explanation of the variables in the list.)

2. If you want to create an isosurface from an existing surface (that is, generate a new surface of constant x, y, temperature, pressure, and so on that is a subset of another surface), choose that surface in the **From Surface** list. You can specify the cell zone on which you want to create an isosurface by selecting the zone in the **From Zones** list.

   - If you do not select a surface from the list, the isosurfacing will be performed on the entire domain.
   - If you do not select a zone from the list, then the isosurfacing will not be restricted to any cell zone and will run through the entire domain.

3. Click **Compute** to calculate the minimum and maximum values of the selected scalar field in the domain or on the selected surface (in the **From Surface** list). The minimum and maximum values will be displayed in the **Min** and **Max** fields.

4. Set the isovalue using one of the following methods. (Note that the second method will enable you to define multiple isovalues in a single isosurface.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You can set an isovalue interactively by moving the slider with the left mouse button. The value in the **Iso-Values** field will be updated automatically. This method will also create a temporary isosurface in the graphics window. Using the slider enables you to preview an isosurface before creating it.

  Important

  Even though the isosurface is displayed, it is only a temporary surface. To create an isosurface, use the **Create** button after deciding on a particular isovalue.

- You can type in isovalues in the **Iso-Values** field directly, separating multiple values by white space. Multiple isovalues will be contained in a single isosurface; that is, you cannot select subsurfaces within the resulting isosurface.

5. If you do not want to use the default name assigned to the surface, enter a new name under **New Surface Name**. The default name is the concatenation of the surface type and an integer that is the new surface ID (for example, `z-coordinate-6`). (If the **New Surface Name** you enter is the same as the name of a surface that already exists, ANSYS Fluent will automatically assign the default name to the new surface when it is created.)

   Important

   The surface name that you enter must begin with an alphabetical letter. If the name of the surface begins with any other character or number, ANSYS Fluent rejects the entry.

6. Click **Create**. The new surface name will be added to the **From Surface** list in the dialog box.

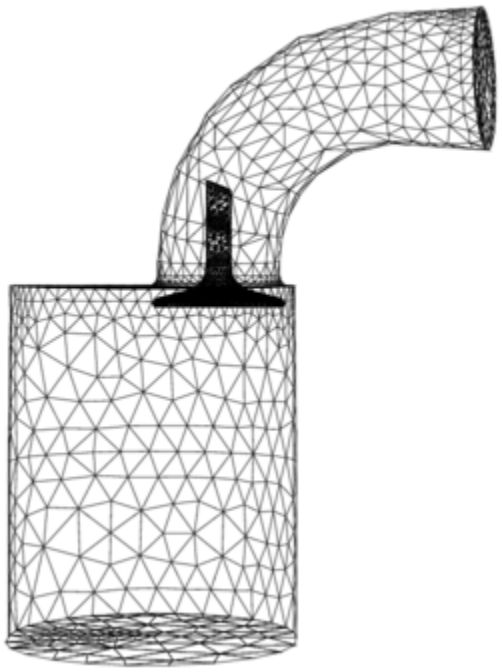
If you want to delete or otherwise manipulate any surfaces, click **Manage...** to open the Surfaces Dialog Box (p. 2810). See Grouping, Renaming, and Deleting Surfaces (p. 1734) for details.

30.1.9. Clipping Surfaces

If you have created a surface, but you do not want to use the whole surface to display data, you can clip the surface between two isovalues to create a new surface that spans a specified subrange of a specified scalar quantity. The clipped surface consists of those points on the selected surface where the scalar field values are within the specified range. For example, in Figure 30.14: External Wall Surface Isoclipped to Values of x Coordinate (p. 1731) the external wall has been clipped to values of x coordinate less than 0 to show only the back half of the wall, enabling you to see the valve inside the intake port.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

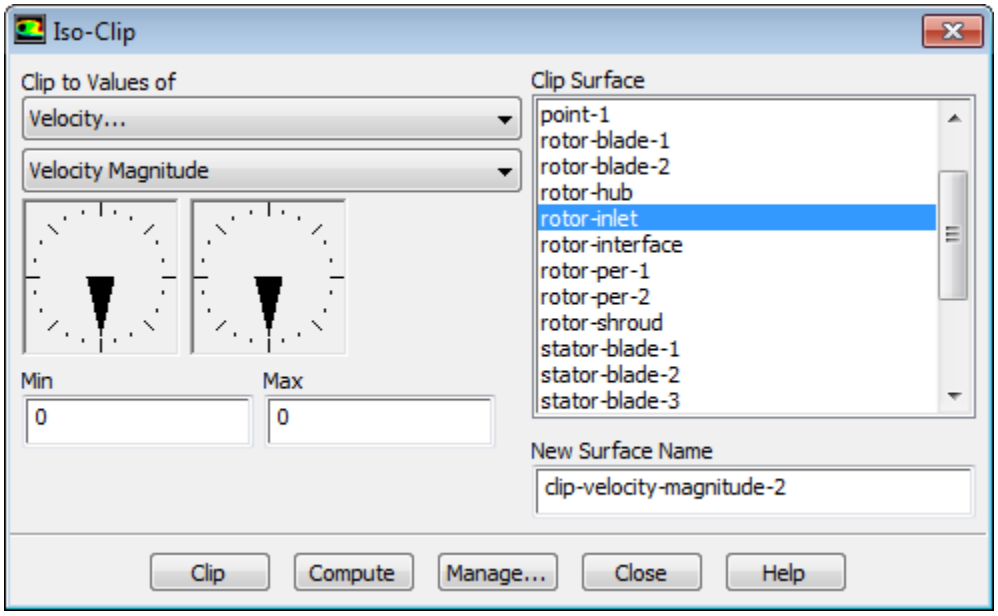
Figure 30.14: External Wall Surface Isoclipped to Values of x Coordinate



To clip an existing surface, you will use the Iso-Clip Dialog Box (p. 2720) (Figure 30.15: The Iso-Clip Dialog Box (p. 1731)).

Setting Up Domain → Surface → Create → Iso-Clip...

Figure 30.15: The Iso-Clip Dialog Box



The steps for clipping a surface are as follows:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Choose the scalar variable on which the clipping will be based in the **Clip To Values Of** drop-down list. First, select the desired category in the upper list. You can then select from related quantities from the lower list. See Field Function Definitions (p. 1931) for an explanation of the variables in the list.

2. Select the surface to be clipped in the **Clip Surface** list.

3. Click **Compute** to calculate the minimum and maximum values of the selected scalar field on the selected surface. The minimum and maximum values are displayed in the **Min** and **Max** fields.

4. Define the clipping range using one of the following methods.

   - You can set the upper and lower limits of the clipping range interactively by moving the indicator in each dial (that is, the dial above the **Min** or **Max** field) with the left mouse button. The value in the corresponding **Min** or **Max** field is updated automatically. This method will also create a temporary surface in the graphics window. Using the dials enables you to preview a clipped surface before creating it.

     Important

     Even though the clipped surface is displayed, it is only a temporary surface. To create the new surface, use the **Clip** button after deciding on the clipping range.

   - You can type the minimum and maximum values in the clipping range directly in the **Min** and **Max** fields.

5. If you do not want to use the default name assigned to the surface, enter a new name under **New Surface Name**. The default name is the concatenation of the surface type and an integer that is the new surface ID (for example, `clip-density-8`). (If the **New Surface Name** you enter is the same as the name of a surface that already exists, ANSYS Fluent will automatically assign the default name to the new surface when it is created.)

   Important

   The surface name that you enter must begin with an alphabetical letter. If the name of the surface begins with any other character or number, ANSYS Fluent rejects the entry.

6. Click **Clip**. The new surface name is added to the **Clip Surface** list in the dialog box. (The original surface will remain unchanged.)

If you want to delete or otherwise manipulate any surfaces, click **Manage...** to open the Surfaces Dialog Box (p. 2810). See Grouping, Renaming, and Deleting Surfaces (p. 1734) for details.

30.1.10. Transforming Surfaces

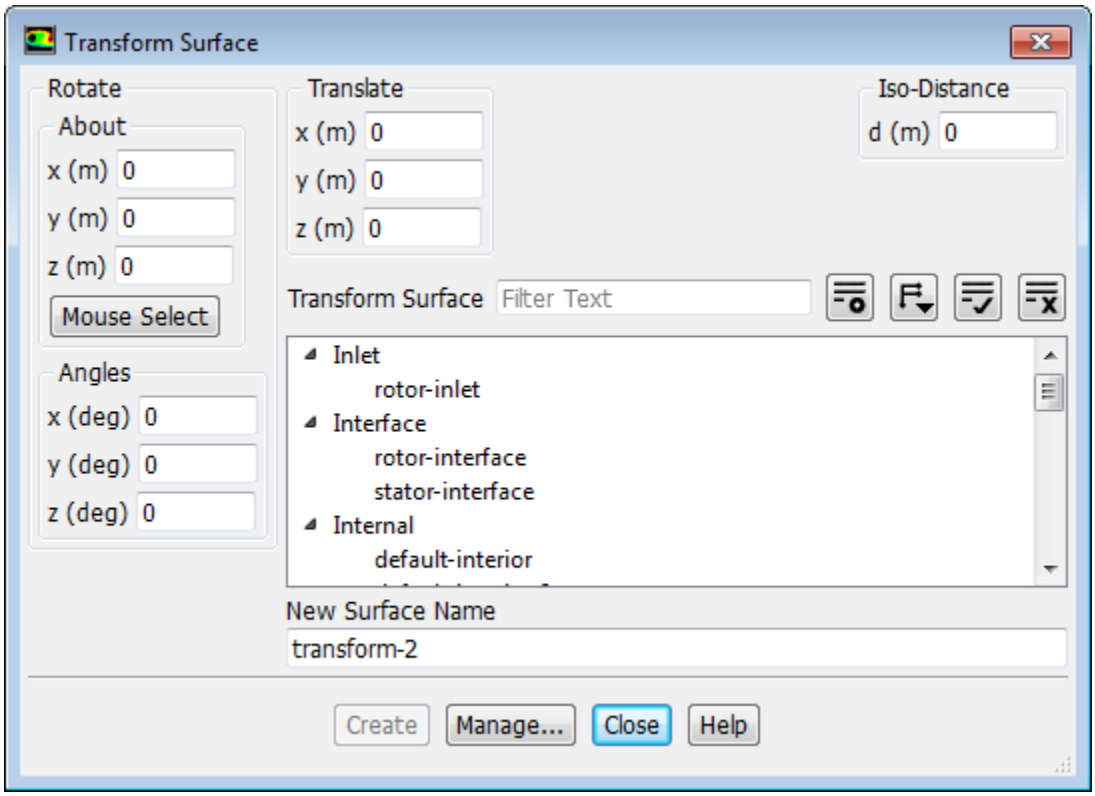
You can create a new data surface from an existing surface by rotating and/or translating the original surface. For example, you can rotate the surface of a complicated turbomachinery blade to plot data in the region between blades. You can also create a new surface at a constant normal distance from the original surface.

To transform an existing surface to create a new one, you will use the Transform Surface Dialog Box (p. 2812) (Figure 30.16: The Transform Surface Dialog Box (p. 1733)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Up Domain → Surface → Create → Transform...

Figure 30.16: The Transform Surface Dialog Box



The steps for transforming a surface are as follows:

1. Select the surface to be transformed in the **Transform Surface** list.

2. Set the appropriate transformation parameters, as described below. You can perform any combination of translation, rotation, and "isodistancing" on the surface.

   - Rotation: To rotate a surface, specify the origin about which the surface is rotated, and the angle by which the surface is rotated.

     In the **About** box under **Rotate**, specify a point, and the origin of the coordinate system for the rotation will be set to that point. (The x, y, and z directions will be the same as for the global coordinate system.) For example, if you specified the point (1,5,3) in 3D, rotation would be about the x, y, and z axes anchored at (1,5,3). You can either enter the point's coordinates in the **x**, **y**, and **z** fields or click **Mouse Select** and select a point in the graphics window using the mouse-probe button. See Controlling the Mouse Button Functions (p. 1800) for information about mouse button functions.

     In the **Angles** box under **Rotate**, specify the angles about the x, y, and z axes (that is, the axes of the coordinate system with the origin defined under **About**) by which the surface is rotated. For 2D problems, you can specify rotation about the z axis only.

   - Translation: To translate a surface, define the distance by which the surface is translated in each direction. Set the **x**, **y**, and **z** translation distances under **Translate**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Isodistancing: To create a surface positioned at a constant normal distance around the original surface, set the value for **d** under **Iso-Distance**.

3. If you do not want to use the default name assigned to the surface, enter a new name under **New Surface Name**. The default name is the concatenation of the surface type and an integer that is the new surface ID (for example, `transform-9`). If the **New Surface Name** you enter is the same as the name of a surface that already exists, ANSYS Fluent will automatically assign the default name to the new surface when it is created.

   Important

   The surface name that you enter must begin with an alphabetical letter. If the name of the surface begins with any other character or number, ANSYS Fluent rejects the entry.

4. Click **Create**. The new surface name is added to the **Transform Surface** list in the dialog box. The original surface will remain unchanged.

If you want to delete or otherwise manipulate any surfaces, click **Manage...** to open the Surfaces Dialog Box (p. 2810). See Grouping, Renaming, and Deleting Surfaces (p. 1734) for details.

30.1.11. Grouping, Renaming, and Deleting Surfaces

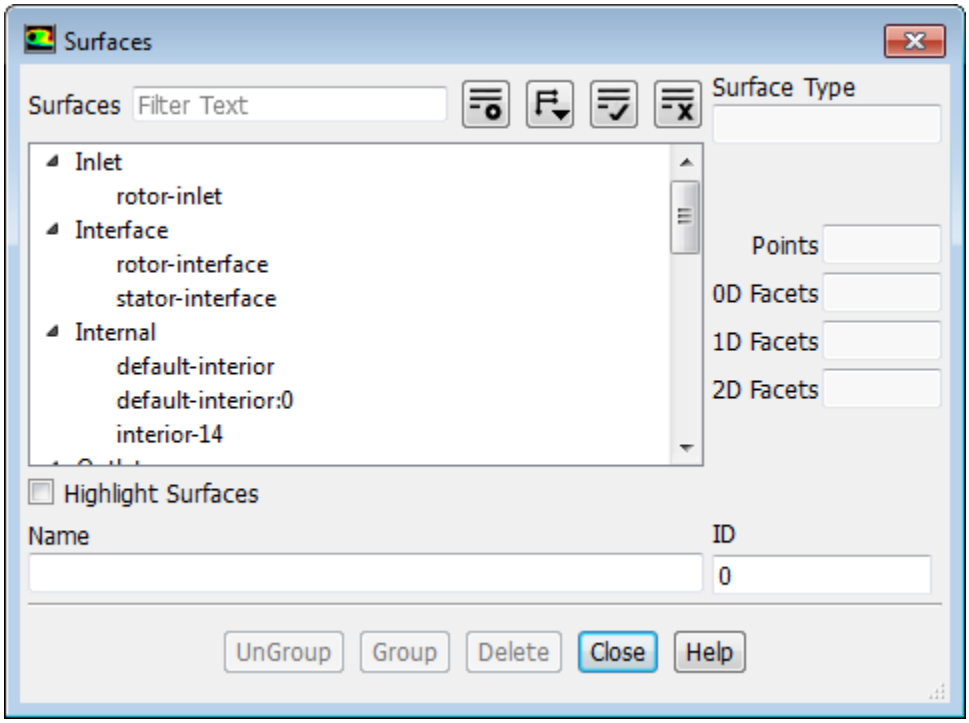
Once you have created a number of surfaces, you can interactively rename, delete, and group surfaces and obtain information about their components. Grouping surfaces is useful if you want to perform postprocessing on a number of surfaces at a time. For example, you may want to group several wall surfaces together to generate a contour plot of temperature on all walls. To postprocess results on each wall surface individually, "ungroup" the surfaces.

Manipulation of existing surfaces is performed with the Surfaces Dialog Box (p. 2810) (Figure 30.17: The Surfaces Dialog Box (p. 1735)).

Setting Up Domain → Surface → Manage...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 30.17: The Surfaces Dialog Box



You can also open this dialog box by clicking **Manage...** in one of the surface creation dialog boxes described in the previous sections.

For additional information, see the following sections:
30.1.11.1. Grouping Surfaces
30.1.11.2. Renaming Surfaces
30.1.11.3. Deleting Surfaces
30.1.11.4. Surface Statistics

30.1.11.1. Grouping Surfaces

As mentioned above, you may want to group several surfaces together in order to perform postprocessing on all of them at once. To create a surface group, select the surfaces to be grouped in the **Surfaces** list. You can define a new name for the group in the **Name** field, or you can use the default name, which is the name of the first surface you selected in the **Surfaces** list. Then click **Group**. The selected surfaces disappear from the **Surfaces** list, and the name of the surface group is added to the list.

Important

The **Group** button will not appear until you have selected at least two surfaces. As soon as you choose a second surface in the **Surfaces** list, the **Rename** button will change to the **Group** button.

To ungroup the surfaces, simply select the surface group in the **Surfaces** list and click **UnGroup**. The group name will disappear from the list and the names of the original surfaces in the group will reappear in the list.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

30.1.11.2. Renaming Surfaces

To change the name of an existing surface, select the surface in the **Surfaces** list, enter a new name in the **Name** field, and then click **Rename**. The new name replaces the old name in the **Surfaces** list and the surface is otherwise unchanged.

If you have selected more than one surface, the **Rename** button will not appear in the dialog box. When more than one surface is selected, the **Rename** button is replaced by the **Group** button.

Important

The surface name that you enter must begin with an alphabetical letter. If the name of the surface begins with any other character or number, ANSYS Fluent rejects the entry.

30.1.11.3. Deleting Surfaces

If you find that a surface is no longer useful, you may want to delete it to prevent the list of surfaces from becoming too cluttered. Select the surface or surfaces to be deleted in the **Surfaces** list, and then click **Delete**. The delete operation is not reversible, so if you want to get a deleted surface back again you will need to recreate it using one of the surface-creation dialog boxes described in the previous sections.

30.1.11.4. Surface Statistics

You can also use the Surfaces Dialog Box (p. 2810) to retrieve topological information about surfaces. **Points** is the total number of nodes in a surface. **0D Facets** is the number of isolated nodes in a surface (that is, nodes that have no connectivity, such as point surfaces or nodes in a rake), **1D Facets** is the number of linear faces (consisting of two connected nodes) in a surface in a 2D problem, and **2D Facets** is the number of 2D faces (triangular or quadrilateral) in a surface in a 3D problem. Note that an **interior** zone surface in a 3D problem consists of 2D facets, and similarly an **interior** zone surface in a 2D problem consists of 1D facets.

These statistics are listed for the surface(s) selected in the **Surfaces** list. If more than one surface is selected, the sum over all selected surfaces is displayed for each quantity.

Note that if you want to check these statistics for a surface that was read from a case file, you will need to first display it.

30.2. Using Cell Registers

The following sections describe how to create and use cell registers:

30.2.1. Region
30.2.2. Boundary
30.2.3. Variable Limiter
30.2.4. Field Variable
30.2.5. Residuals
30.2.6. Volume
30.2.7. Yplus/Ystar
30.2.8. Manage Cell Registers
30.2.9. Cell Register Operations

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

30.2.1. Region

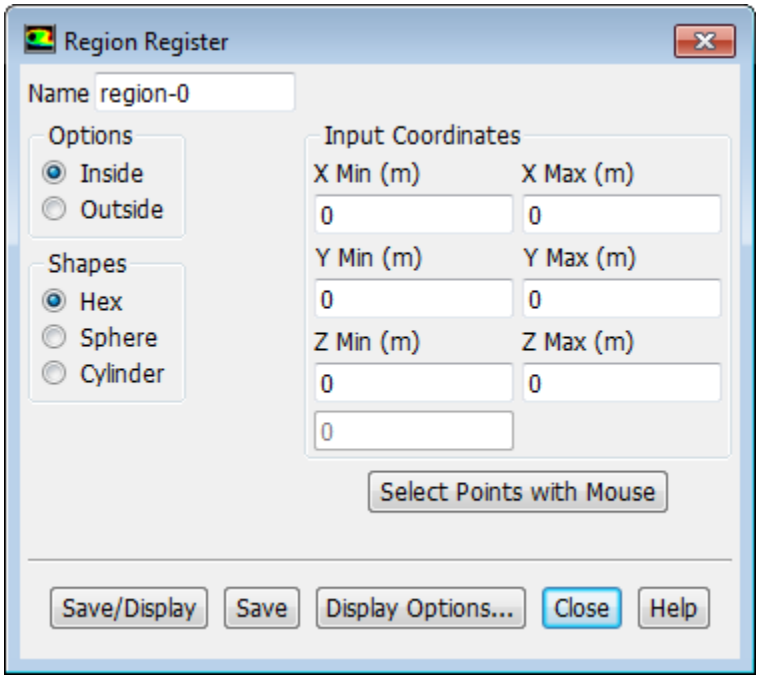
Region registers allow you to mark cells inside or outside of a given region that you can specify using text or mouse inputs. Using text input, you can specify the region as a hexahedron (quadrilateral in 2D), a sphere (circle in 2D), or a cylinder. For additional information on region cell registers, see Region Adaption in the *Fluent Theory Guide*.

To create a region cell register:

1. Open the Figure 30.18: Region Register Dialog Box (p. 1737).

   Solution → **Cell Registers** **New** → **Region...**

   Figure 30.18: Region Register Dialog Box



2. Specify whether the register includes cells **Inside** or **Outside** of the defined region.

3. Choose the shape for the region register.

4. Specify the coordinates of the region.

   You can either enter the X, Y, and Z values manually to create the region, or you can click **Select Points with Mouse** to use the mouse-probe to select the points directly on the model in the graphics window.

5. Click **Save** to create the cell register.

Note

You can change the way the register is displayed by using the **Cell Register Display Options** dialog box that opens by clicking **Display Options...**. The controls in the **Cell**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Register Display Options dialog box are similar to those in the Adaption Display Options Dialog Box (p. 2663).

30.2.2. Boundary

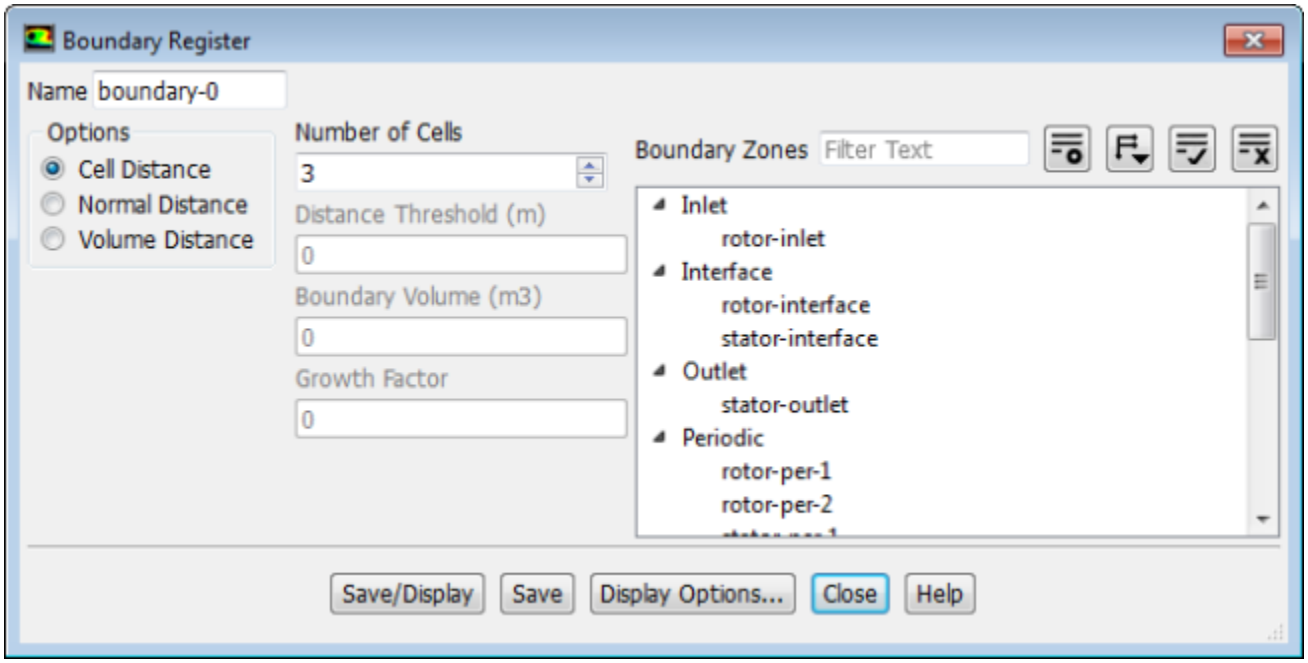
Boundary registers allow you to mark cells within the proximity of a boundary. The proximity to the boundary can be based on cell distance, normal distance, or volume distance. For additional information on boundary registers, see Boundary Adaption in the *Fluent Theory Guide*.

To create a boundary cell register:

1. Open the Figure 30.19: Boundary Register Dialog Box (p. 1738).

   Solution → **Cell Registers** **New** → **Boundary...**

   Figure 30.19: Boundary Register Dialog Box



2. Choose whether the boundary register is based on **Cell Distance**, **Normal Distance**, or **Volume Distance**.
3. Specify the **Number of Cells**, **Distance Threshold**, or **Boundary Volume** and **Growth Factor**, depending on which distance method you selected in the previous step.
4. Select the boundary zone(s) for the cell register.
5. Click **Save** to create the cell register.

Note

You can change the way the register is displayed by using the **Cell Register Display Options** dialog box that opens by clicking **Display Options...**. The controls in the **Cell**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Register Display Options dialog box are similar to those in the Adaption Display Options Dialog Box (p. 2663).

30.2.3. Variable Limiter

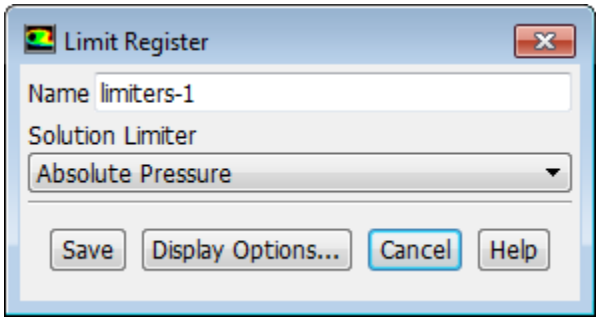
Limit registers allow you to mark the cells that have reached a solution limit while solving. These limits can be viewed and modified in Figure 28.11: The Solution Limits Dialog Box (p. 1586). For additional information on solution limits, see Setting Solution Limits (p. 1586).

To create a limit cell register:

1. Open the Figure 30.20: Limit Register Dialog Box (p. 1739).

   Solution → Cell Registers **New → Variable Limiter...**

   Figure 30.20: Limit Register Dialog Box

2. Select the variable for creating the cell register.
3. Click **Save** to create the cell register.

Note

You can change the way the register is displayed by using the **Cell Register Display Options** dialog box that opens by clicking **Display Options...**. The controls in the **Cell Register Display Options** dialog box are similar to those in the Adaption Display Options Dialog Box (p. 2663).

30.2.4. Field Variable

Field variable registers allow you to mark cells based on the value of a field variable. You can mark cells based on the minimum/maximum value of a field variable, inside or outside of a certain range of field variable values, or within a certain percentage of the minimum/maximum value of a field variable.

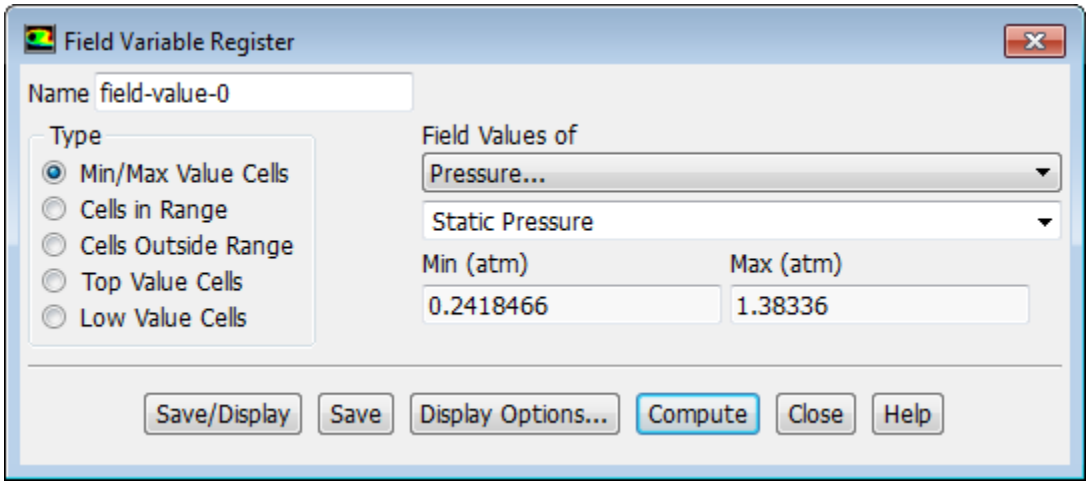
To create a field variable cell register:

1. Open the Figure 30.21: Field Variable Register Dialog Box (p. 1740).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution → Cell Registers → New → Field Variable...

Figure 30.21: Field Variable Register Dialog Box



2. Choose the **Type** of field variable register. Note that you can use **Compute** to calculate the minimum and maximum values of the chosen field variable.
3. Select the field variable for creating the cell register from the **Field Values of** drop-down lists.
4. Click **Save** to create the cell register.

Note

You can change the way the register is displayed by using the **Cell Register Display Options** dialog box that opens by clicking **Display Options...**. The controls in the **Cell Register Display Options** dialog box are similar to those in the Adaption Display Options Dialog Box (p. 2663).

30.2.5. Residuals

Residual registers allow you to mark cells based on their residuals values. You can visualize the cells with the highest residual values and know their location so that you can modify them as necessary.

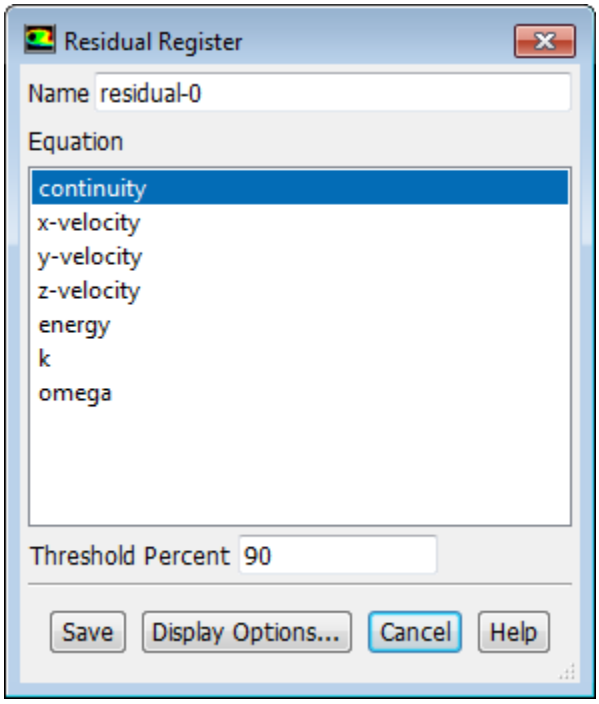
To create a residual cell register:

1. Open the Figure 30.22: Residual Register Dialog Box (p. 1741).

Solution → Cell Registers → New → Residuals...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 30.22: Residual Register Dialog Box



2. Select the **Equation**(s) for creating the residual cell register.

3. Specify the **Threshold Percent** for creating the residual cell register. Cells with residual values above this threshold for the selected equations are marked.

4. Click **Save** to create the cell register.

Note

You can change the way the register is displayed by using the **Cell Register Display Options** dialog box that opens by clicking **Display Options...**. The controls in the **Cell Register Display Options** dialog box are similar to those in the Adaption Display Options Dialog Box (p. 2663).

30.2.6. Volume

Volume registers allow you to mark cells based on the volume of the cells, either the magnitude or the change in volume. For additional information on volume cell registers, see Volume Adaption in the *Fluent Theory Guide*.

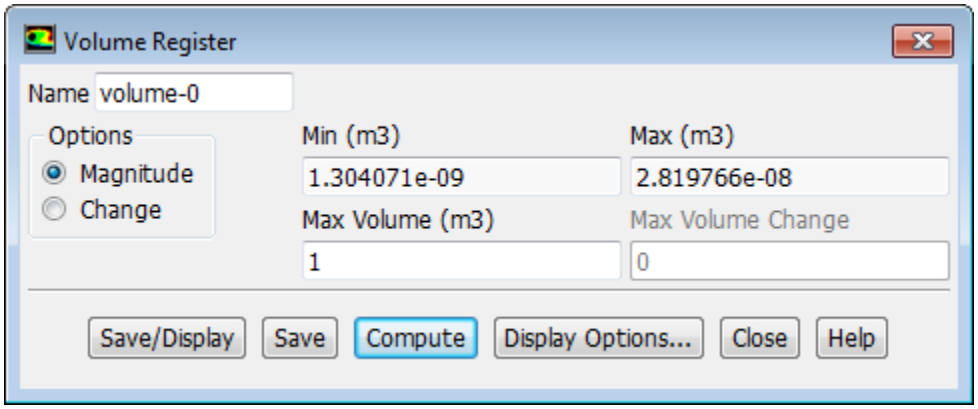
To create a volume cell register:

1. Open the Figure 30.23: Volume Register Dialog Box (p. 1742).

Solution → Cell Registers → New → Volume...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 30.23: Volume Register Dialog Box



2. Choose whether the cell register is based on the maximum size of the cells (**Magnitude**) or the **Change** in volume of the cells.

3. Specify the **Max Volume** or **Max Volume Change**. Note that you can click **Compute** to compute the minimum and maximum cell volumes.

4. Click **Save** to create the cell register.

Note

You can change the way the register is displayed by using the **Cell Register Display Options** dialog box that opens by clicking **Display Options...**. The controls in the **Cell Register Display Options** dialog box are similar to those in the Adaption Display Options Dialog Box (p. 2663).

30.2.7. Yplus/Ystar

Yplus/Ystar registers allow you to mark cells based on their Yplus/Ystar values. For additional information on Yplus/Ystar cell registers, see Yplus/Ystar Adaption in the *Fluent Theory Guide*.

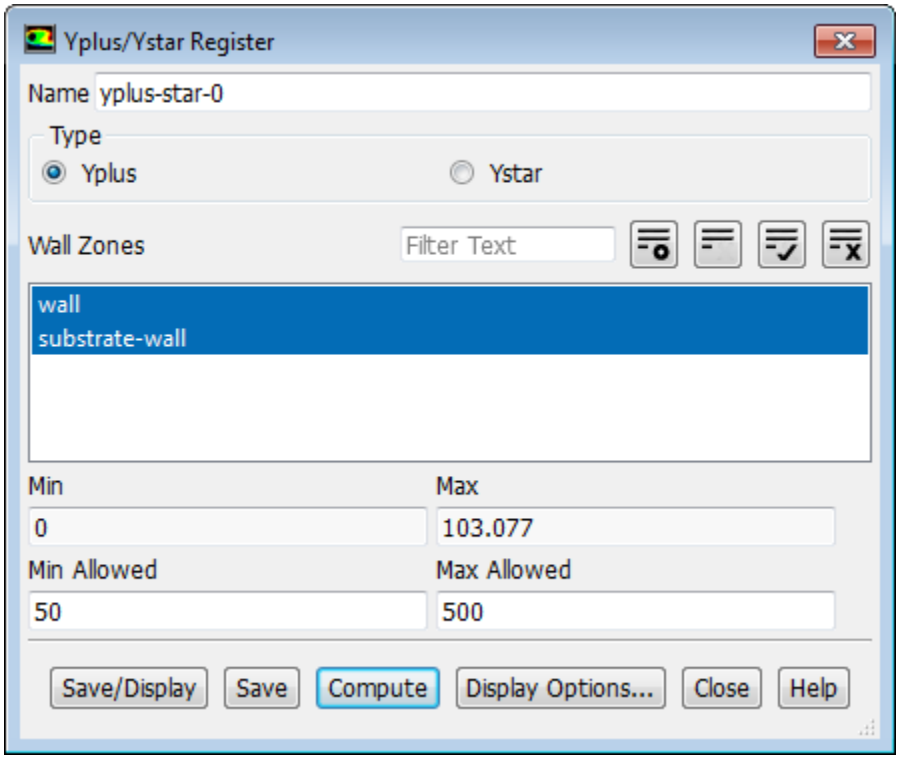
To create a Yplus/Ystar cell register:

1. Open the Figure 30.24: Yplus/Ystar Register Dialog Box (p. 1743).

   Solution → **Cell Registers** → **New** → **Yplus/Ystar...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 30.24: Yplus/Ystar Register Dialog Box



2. Choose whether the cell register is based on **Yplus** or **Ystar** values.
3. Select the **Wall Zones** for the Yplus or Ystar calculations.
4. Specify the minimum and maximum Yplus/Ystar values allowed. Note that clicking **Compute** calculates the minimum and maximum Yplus/Ystar values.
5. Click **Save** to create the cell register.

Note

You can change the way the register is displayed by using the **Cell Register Display Options** dialog box that opens by clicking **Display Options...**. The controls in the **Cell Register Display Options** dialog box are similar to those in the Adaption Display Options Dialog Box (p. 2663).

30.2.8. Manage Cell Registers

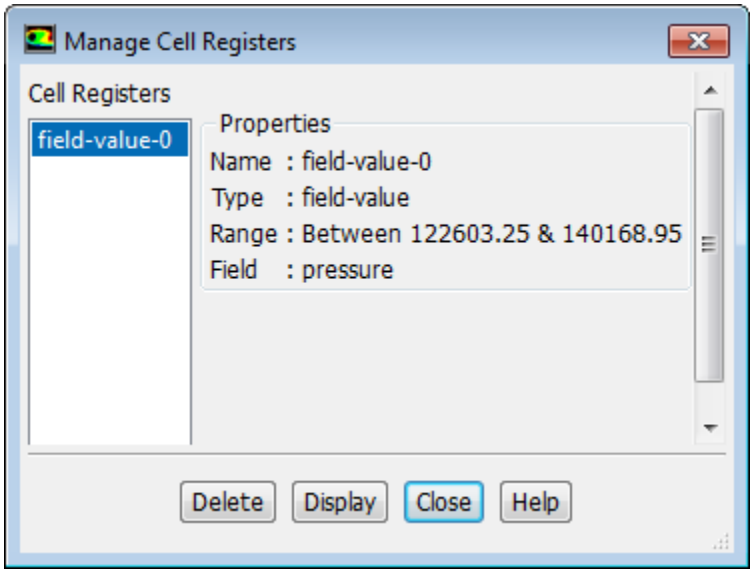
The Figure 30.25: Manage Cell Registers Dialog Box (p. 1744) allows you to review the properties of the currently defined cell registers. You can also use it to display and delete the selected cell register.

To manage cell registers:

1. Open the Figure 30.25: Manage Cell Registers Dialog Box (p. 1744).

   Solution → **Cell Registers** **Edit...**

Figure 30.25: Manage Cell Registers Dialog Box



2. Select a cell register to: view its properties, display it, or delete it.

30.2.9. Cell Register Operations

Once a cell register is defined, you can create a cell register operation to report the residual values of the selected cell registers at the interval you specify.

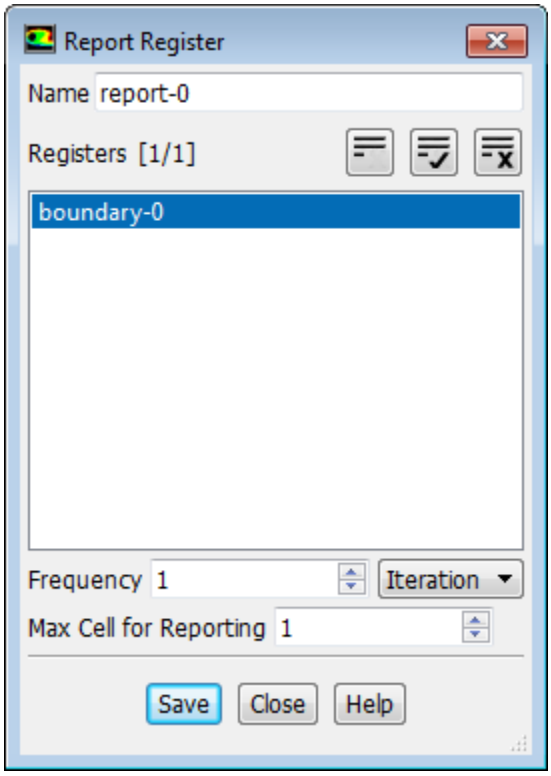
To create a register report:

1. Open the Figure 30.26: Report Register Dialog Box (p. 1745).

Solution → Calculation Activities → Cell Register Operations → New Report...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 30.26: Report Register Dialog Box



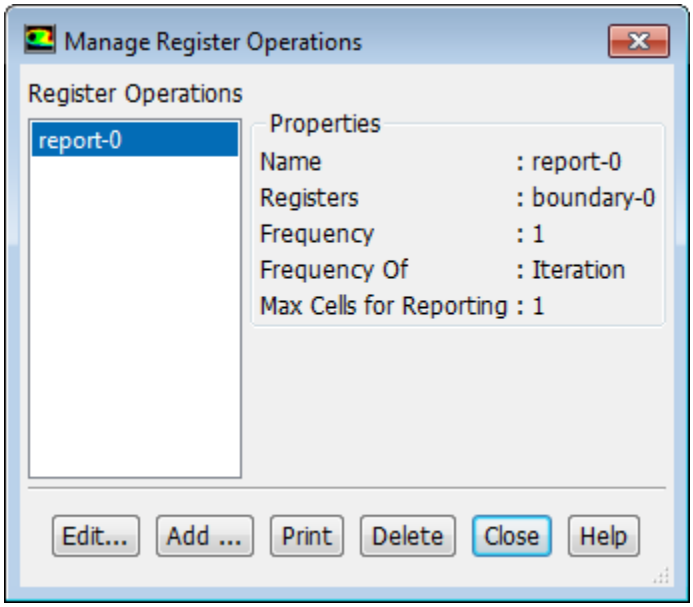
Note that this dialog box is only available after you have defined a cell register.

2. Select the cell register(s) you want included in the report.
3. Specify the reporting **Frequency**.
4. Specify the maximum number of cells to include in the report.
5. Click **Save** to create the cell register report.

The Figure 30.27: Manage Register Operations Dialog Box (p. 1746) allows you to view properties, create, modify, and delete report registers.

Solution → Calculation Activities → Cell Register Operations Edit...

Figure 30.27: Manage Register Operations Dialog Box



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 31: Displaying Graphics

Graphics tools available in ANSYS Fluent enable you to process the information contained in your CFD solution and easily view the results. The following sections explain how to use these tools to examine your solution. The procedure for saving picture files of graphics displays is described in Saving Picture Files (p. 102).

31.1. Basic Graphics Generation
31.2. Customizing the Graphics Display
31.3. Controlling the Mouse Button Functions
31.4. Viewing the Application Window
31.5. Modifying the View
31.6. Advanced Scene Composition
31.7. Animating Graphics
31.8. Creating Videos
31.9. Histogram and XY Plots
31.10. Turbomachinery Postprocessing
31.11. Fast Fourier Transform (FFT) Postprocessing

31.1. Basic Graphics Generation

In ANSYS Fluent, you can generate graphics displays showing meshes, contours, profiles, vectors, pathlines, and scenes. Some graphics are generated using variables that are plotted directly from the ANSYS Fluent data file once the file has been read. The variables listed in the data file depend on the models active at the time the file is written. Variables that are required by the solver, based on the current model settings, but are missing from the data file are set to their default values. For those missing variables, one iteration should be performed in order to obtain the required values for generating the plot. A complete list of variables stored in the data file is available in (`xfile.h`) and can be accessed as stated in Data Sections (p. 2865). The following sections describe how to create these plots. (Generation of histogram and XY plots is discussed in Histogram and XY Plots (p. 1838).)

Important

If your model includes a discrete phase, you can also display the particle trajectories, as described in Displaying of Trajectories (p. 1330).

Note

There are two kinds of graphics objects—those whose settings are saved with the case file for reuse during the simulation, and those that are not. Persistent graphics objects (saved with the case file) are accessed by clicking **New...**, non-persistent objects are created by clicking **Edit...**. For example, you can create a persistent contour plot by clicking **Contours** and selecting **New...** in the **Postprocessing** ribbon tab (**Graphics** group box).

Postprocessing → **Graphics** → **Contours** → **New...**

This section discusses the following topics:

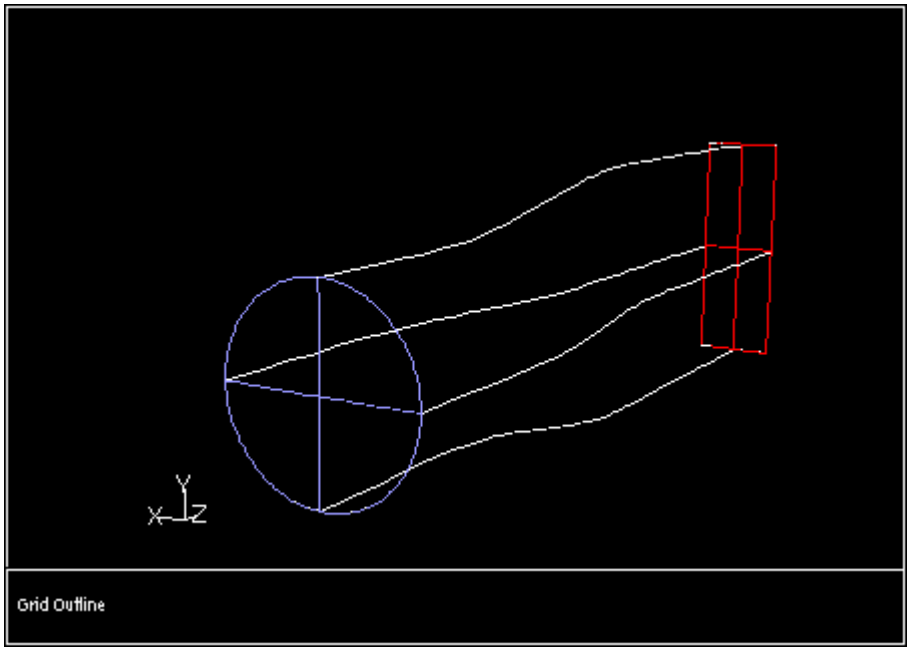
31.1.1. Displaying the Mesh
31.1.2. Displaying Contours and Profiles
31.1.3. Displaying Vectors
31.1.4. Displaying Pathlines
31.1.5. Displaying a Scene
31.1.6. Displaying Results on a Sweep Surface
31.1.7. Hiding the Graphics Window Display

31.1.1. Displaying the Mesh

During the problem setup or when you are examining your solution, you may want to look at the mesh associated with certain surfaces. You can do the following:

- Display the outline of all or part of the domain, as shown in Figure 31.1: Outline Display (p. 1748).

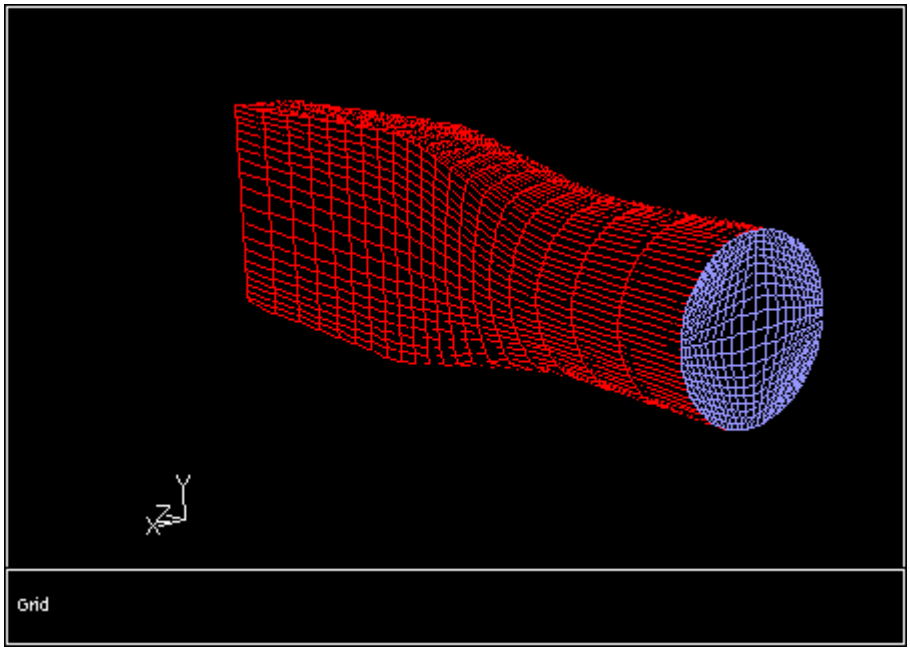
Figure 31.1: Outline Display



- Draw the mesh lines (edges), as shown in Figure 31.2: Mesh Edge Display (p. 1749).

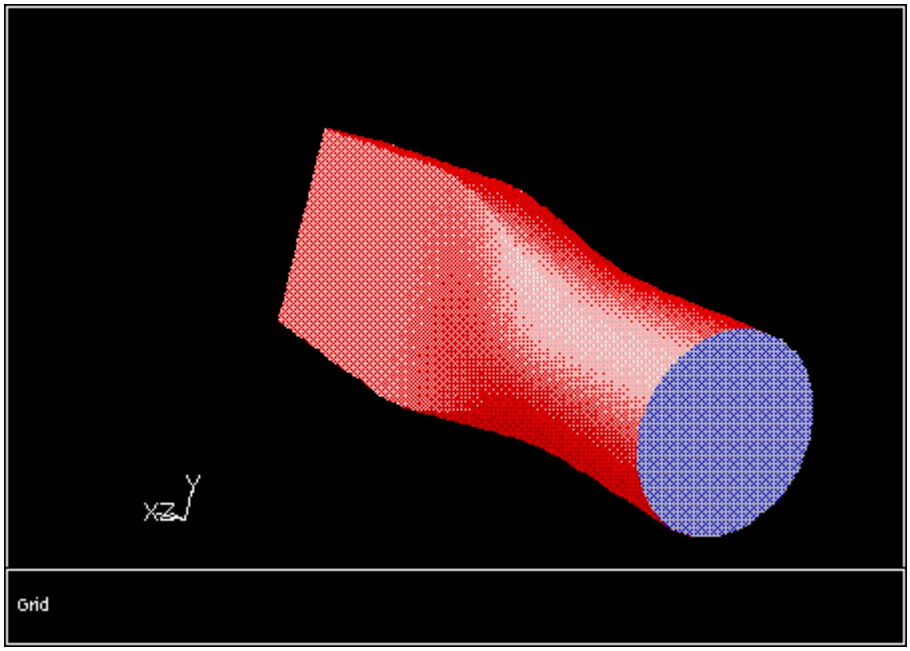
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.2: Mesh Edge Display



- Draw the solid surfaces (filled meshes) for a 3D domain, as shown in Figure 31.3: Mesh Face (Filled Mesh) Display (p. 1749).

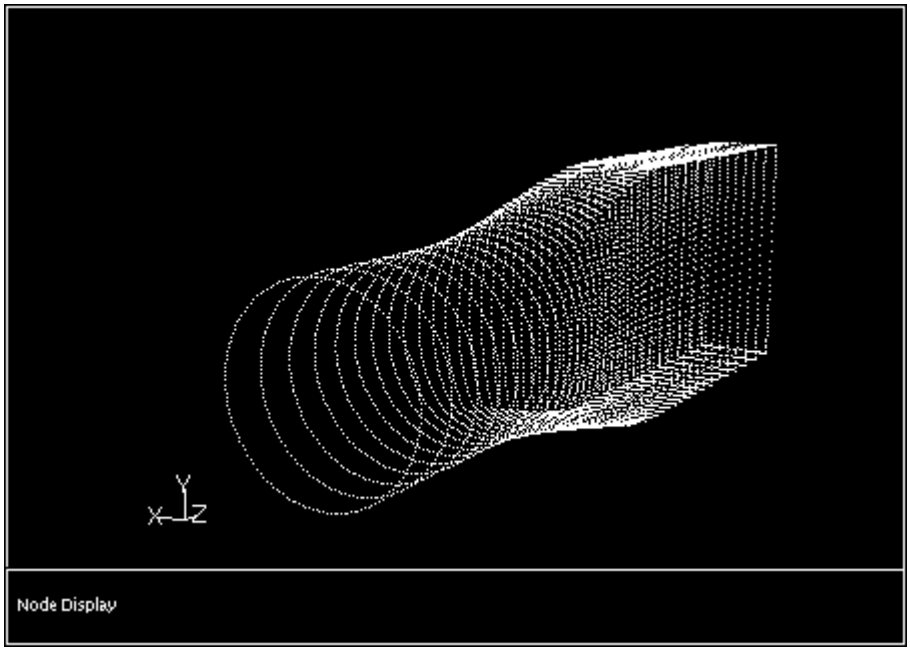
Figure 31.3: Mesh Face (Filled Mesh) Display



- Draw the nodes on the domain surfaces, as shown in Figure 31.4: Node Display (p. 1750).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.4: Node Display



Note

The **Post Processing** option for mesh display configuration is recommended when postprocessing your results in ANSYS Fluent. This option is accessed by clicking in the objects toolbar or by entering `dislay/set/mesh-display-configuration` in the console.

For information about displaying the mesh on a surface that sweeps through the domain, see Displaying Results on a Sweep Surface (p. 1782).

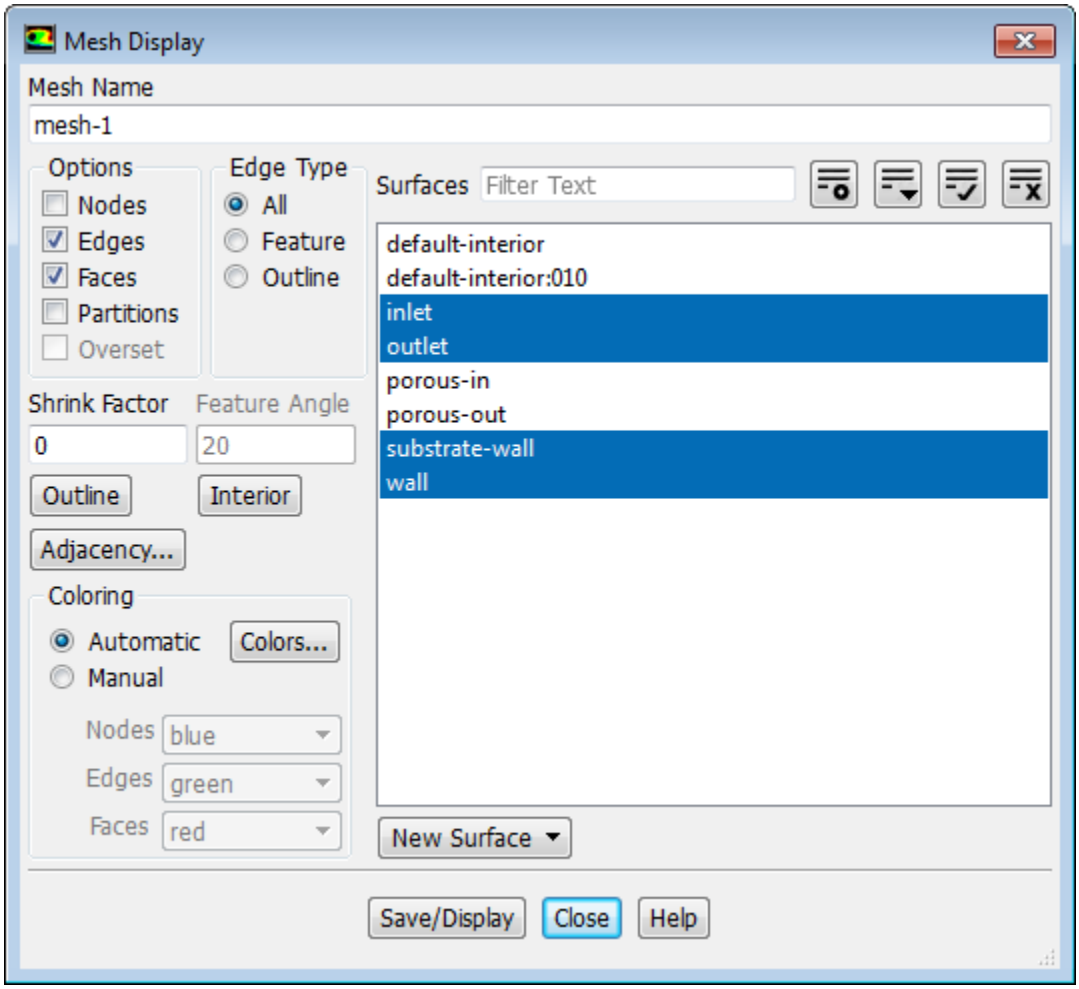
31.1.1.1. Generating Mesh or Outline Plots

You can draw the mesh or outline for all or part of your domain using the Mesh Display Dialog Box (p. 2163) (Figure 31.5: The Mesh Display Dialog Box (p. 1751)).

Results → Graphics → Mesh Edit...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.5: The Mesh Display Dialog Box



The basic steps for generating a mesh or outline plot are as follows:

1. Choose the surfaces for which you want to display the mesh or outline in the **Surfaces** list.

   If you want to select several surfaces of the same type, click and select **Surface Type** under **Group By**, which organizes the surfaces in a tree view that is grouped by surface type.

   Another shortcut is to use the **Filter Text** entry box to filter the **Surfaces** list to show only the surfaces that match the pattern you enter. For additional information on using the **Filter Text** entry box, see Filter Text Entry Boxes (p. 15).

2. Depending on what you want to draw, do one or more of the following:

   - To draw an outline of the selected surfaces (as in Figure 31.1: Outline Display (p. 1748)), select **Edges** under **Options** and **Outline** under **Edge Type**. If you need more detail in the outline display of a complex geometry, see the description of the **Feature** option, below.

   - To draw the mesh edges (as in Figure 31.2: Mesh Edge Display (p. 1749)), select **Edges** under **Options** and **All** under **Edge Type**.

   - To generate a filled-mesh display (as in Figure 31.3: Mesh Face (Filled Mesh) Display (p. 1749)), select **Faces** under **Options**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To draw the nodes on the selected surfaces (as in Figure 31.4: Node Display (p. 1750)), select **Nodes** under **Options**.

3. Set any of the mesh and outline display options described in the following section.

4. Click **Display** to draw the specified mesh or outline in the active graphics window.

To display filled meshes, with smoothly shaded display, enable lighting and select a lighting interpolation method other than **Flat** in the Display Options Dialog Box (p. 2573) or the Lights Dialog Box (p. 2588).

If you display nodes, and you want to change the symbol representing the nodes, you can change the **Point Symbol** in the Display Options Dialog Box (p. 2573). See Modifying the Rendering Options (p. 1798) for details.

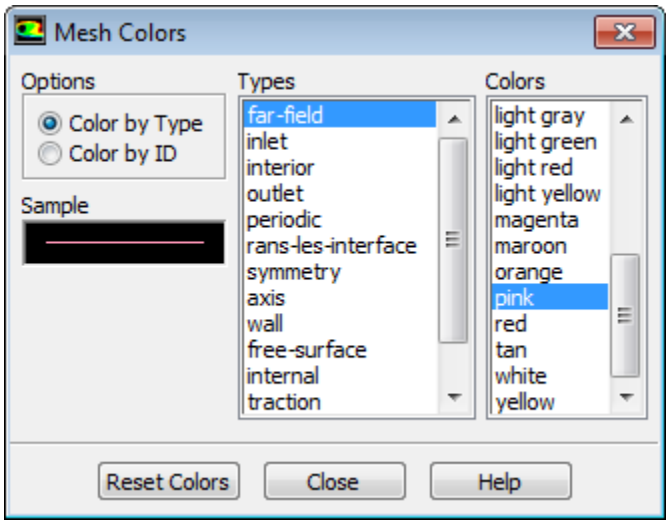
31.1.1.2. Mesh and Outline Display Options

The options mentioned in the procedure in the previous section include modifying the mesh colors, adding the outline of important features to an outline display, drawing partition boundaries, and shrinking the faces and/or cells in the display.

31.1.1.2.1. Modifying the Mesh Colors

ANSYS Fluent enables you to control the colors that are used to render the meshes for each zone type or surface. This capability can help you to understand mesh plots quickly and easily. To modify the colors, open the Mesh Colors Dialog Box (p. 2168) (Figure 31.6: The Mesh Colors Dialog Box (p. 1752)) by clicking **Colors...** in the Mesh Display Dialog Box (p. 2163).

Figure 31.6: The Mesh Colors Dialog Box



You can set colors individually for the meshes displayed on each surface, using the Scene Description Dialog Box (p. 2577).

By default, the **Color by Type** option is turned on, enabling you to assign colors based on zone type. To change the color used to draw the mesh for a particular zone type, select the zone type in the **Types** list and then select the new color in the **Colors** list. You will see the effect of your change when you next display the mesh. Note that the **surface** type in the **Types** list applies to all surface meshes (that

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

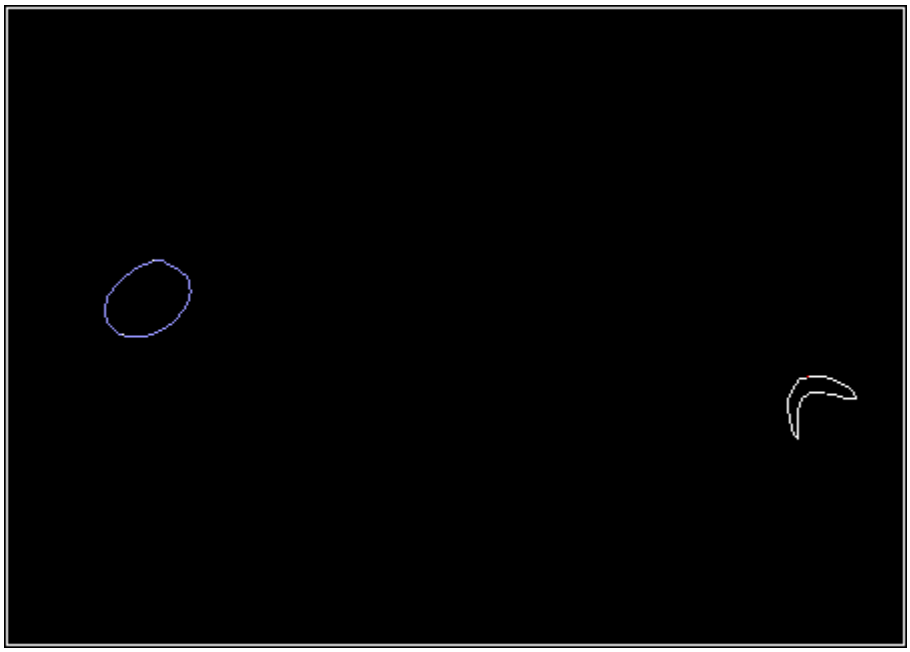
is, meshes that are drawn for surfaces created using the dialog boxes opened from the **Create** button in the **Surface** group box of the **Setting Up Domain** ribbon tab) except zone surfaces.

If you prefer to use the colors ANSYS Fluent assigns by zone ID, then you can display the mesh using the **Color by ID** option.

31.1.1.2.2. Adding Features to an Outline Display

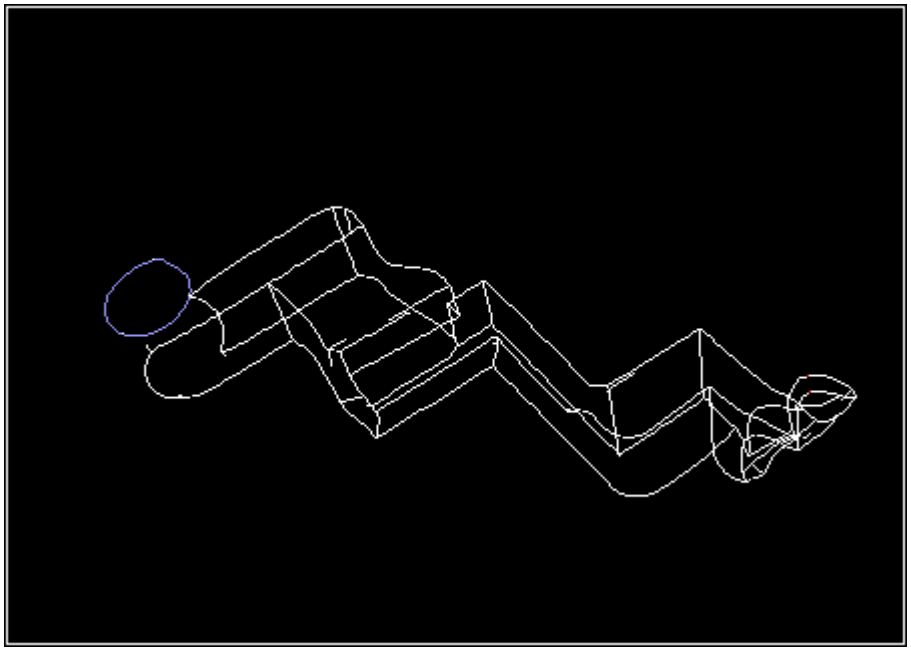
For closed 3D geometries such as cylinders, the standard outline display often will not show enough detail to accurately depict the shape. This is because for each boundary, only those edges on the "outside" of the geometry (that is, those that are used by only one face on the boundary) are drawn. In Figure 31.7: Standard Outline of Complex Duct (p. 1753), which shows the outline display for a complicated duct geometry, only the inlet and outlet are visible.

Figure 31.7: Standard Outline of Complex Duct



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.8: Feature Outline of Complex Duct



You can capture additional features using the **Feature** option in the Mesh Display Dialog Box (p. 2163). (See Figure 31.8: Feature Outline of Complex Duct (p. 1754).) Enable **Feature** under **Edge Type**, and then set the **Feature Angle**. With the default **Feature Angle** of 20, if the difference between the normal directions of two adjacent faces is more than 20°, the edge between those faces will be drawn. Decreasing the **Feature Angle** will result in more edge lines (that is, more detail) being added to the outline display. The appropriate angle for your geometry will depend on its curvature and complexity. You can modify the **Feature Angle** until you find the value that yields the best outline display.

31.1.1.2.3. Drawing Partition Boundaries

If you have partitioned your mesh for parallel processing, you can add the display of partition boundaries to the mesh display by turning on the **Partitions** option in the Mesh Display Dialog Box (p. 2163).

31.1.1.2.4. Shrinking Faces and Cells in the Display

To distinguish individual faces or cells in the display, enlarge the space between adjacent faces or cells by increasing the **Shrink Factor** in the Mesh Display Dialog Box (p. 2163). The default value of zero produces a display in which the edges of adjacent faces or cells overlap. A value of 1 creates the opposite extreme: each face or cell is represented by a point and there is considerable space between each one. A small value such as 0.01 may be large enough to enable you to distinguish one face or cell from its neighbor. Displays with different **Shrink Factor** values are shown in Figure 31.9: Mesh Display with Shrink Factor = 0 (p. 1755) and Figure 31.10: Mesh Display with Shrink Factor = 0.01 (p. 1755). Click **Display** to see the effect of the change in **Shrink Factor**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.9: Mesh Display with Shrink Factor = 0

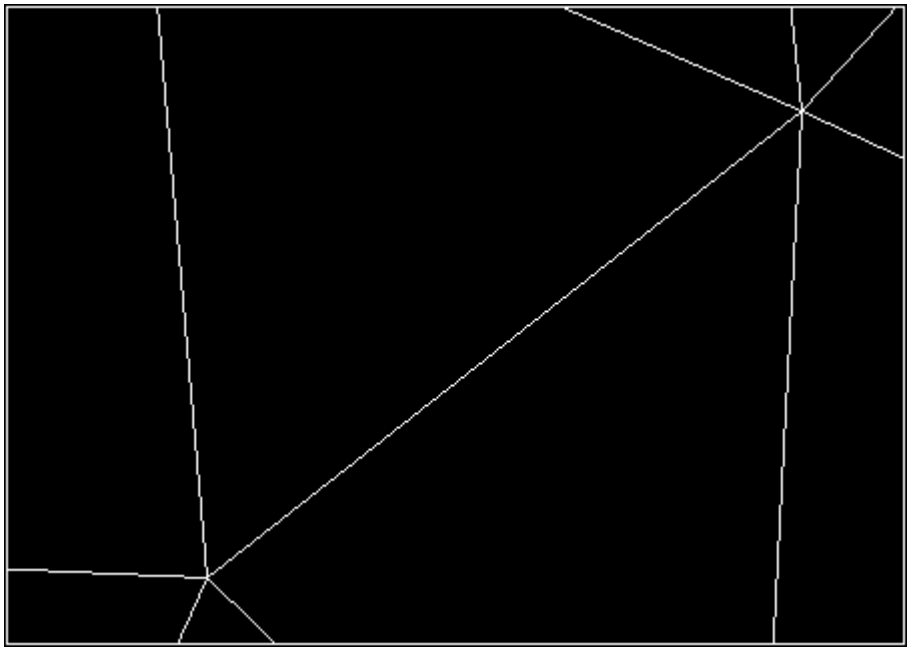
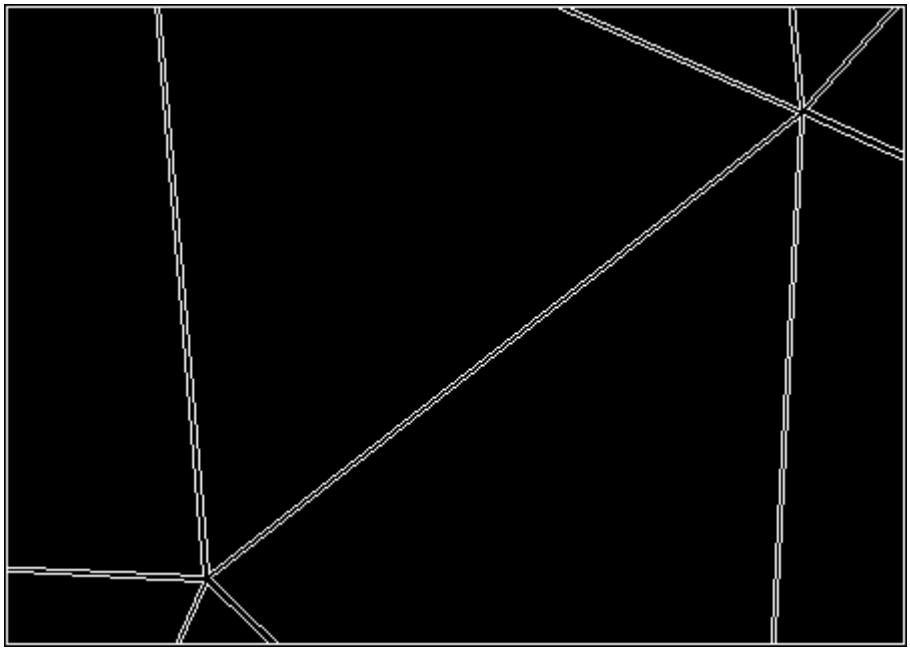


Figure 31.10: Mesh Display with Shrink Factor = 0.01



31.1.1.3. Creating and Using Mesh Plot Definitions

You can create named mesh plot definitions and save them for later use. See Creating and Using Contour Plot Definitions (p. 1762) for additional information on graphics object definitions.

The steps for creating and using mesh plot definitions are similar to the steps for Generating Mesh or Outline Plots (p. 1750), with the addition of a field for naming the mesh plot definition.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.1.2. Displaying Contours and Profiles

ANSYS Fluent enables you to plot contour lines or profiles superimposed on the physical domain. Contour lines are lines of constant magnitude for a selected variable (isotherms, isobars, and so on). A profile plot draws these contours projected off the surface along a reference vector by an amount proportional to the value of the plotted variable at each point on the surface. Sample plots are shown in Figure 31.11: Contours of Static Pressure (p. 1756) and Figure 31.12: Profile Plot of y Velocity (p. 1756).

For information about displaying contours or profiles on a surface that sweeps through the domain, see Displaying Results on a Sweep Surface (p. 1782).

Figure 31.11: Contours of Static Pressure

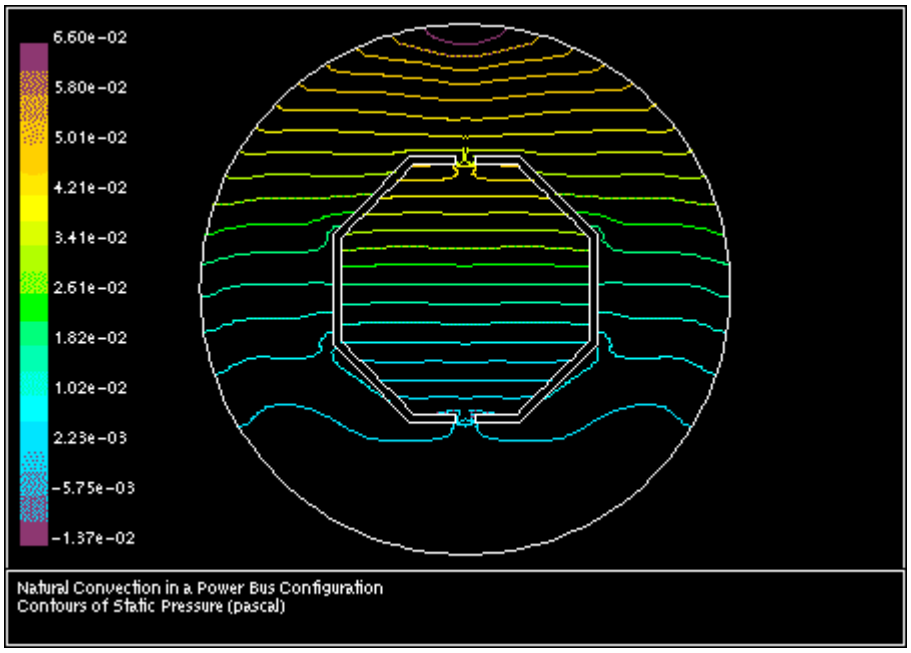
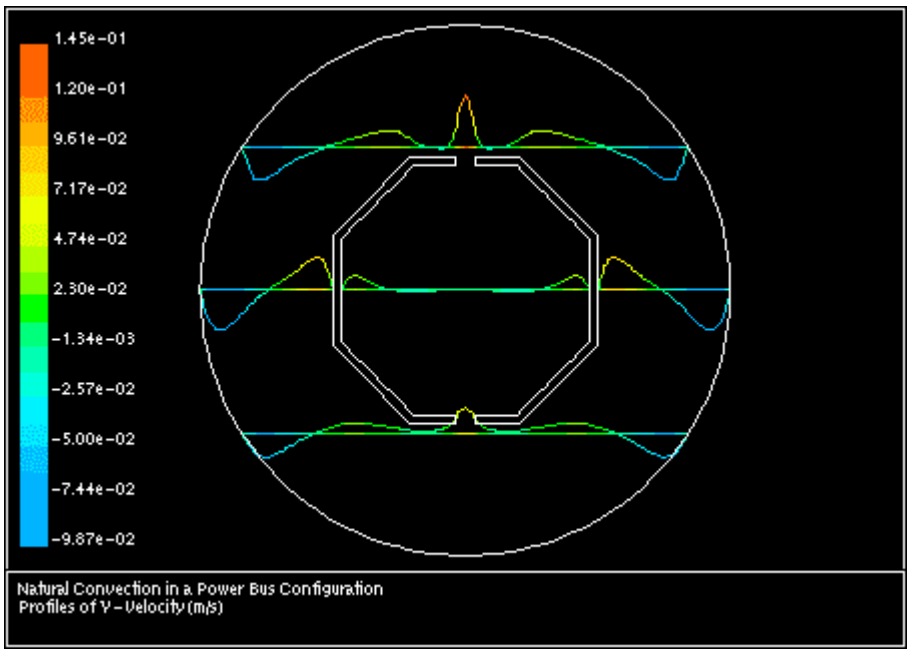


Figure 31.12: Profile Plot of y Velocity



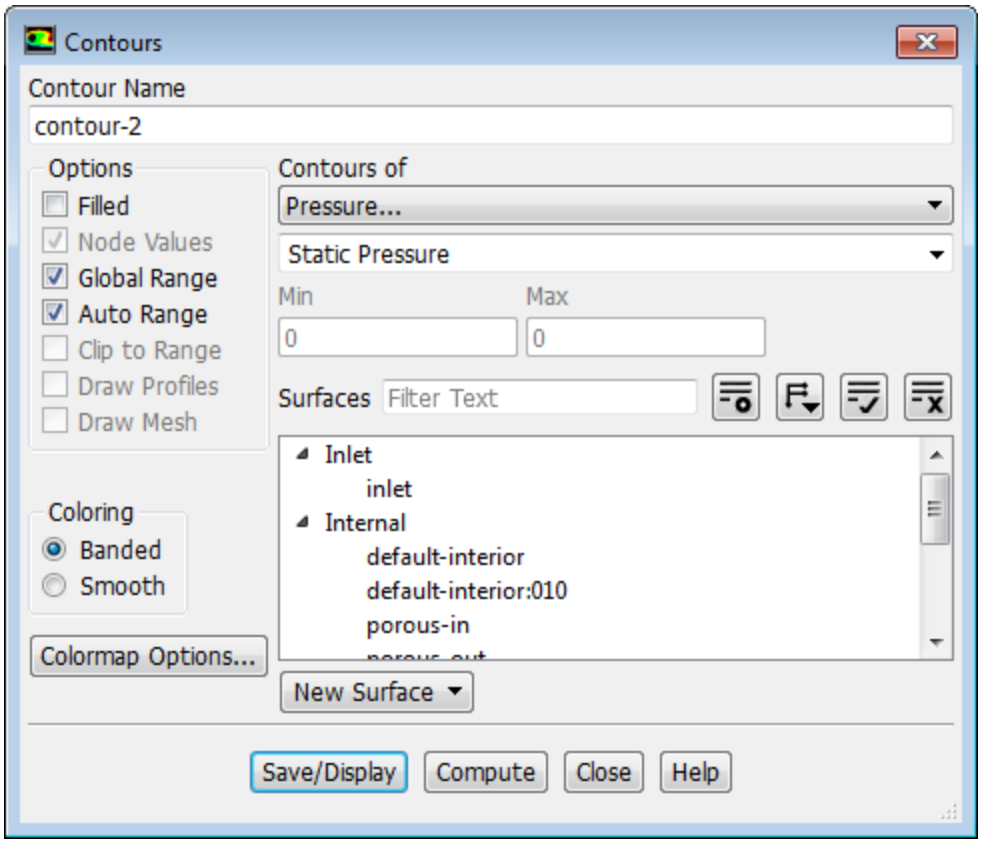
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.1.2.1. Generating Contour and Profile Plots

You can plot contours or profiles using the Contours Dialog Box (p. 2676) (Figure 31.13: The Contours Dialog Box (p. 1757)).

Results → Graphics → Contours Edit...

Figure 31.13: The Contours Dialog Box



The basic steps for generating a contour or profile plot are as follows:

1. Select the variable or function to be contoured or profiled in the **Contours of** drop-down list. First select the desired category in the upper list; you may then select a related quantity in the lower list. (See Field Function Definitions (p. 1931) for an explanation of the variables in the list.)

2. Choose the surface or surfaces on which to draw the contours or profiles in the **Surfaces** list. For 2D cases, if no surface is selected, contouring or profiling is done on the entire domain. For 3D cases, you must always select at least one surface.

   If you want to select several surfaces of the same type, click and select **Surface Type** under **Group By**, which organizes the surfaces in a tree view that is grouped by surface type.

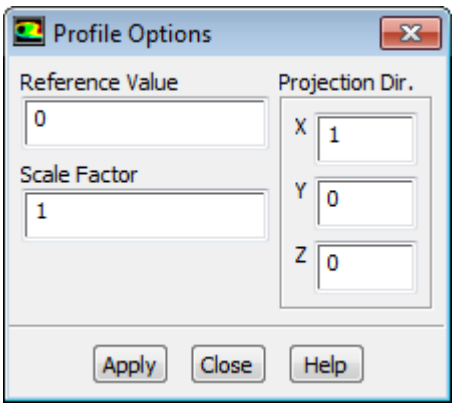
   Another shortcut is to use the **Filter Text** entry box to filter the **Surfaces** list to show only the surfaces that match the pattern you enter. For additional information on using the **Filter Text** entry box, see Filter Text Entry Boxes (p. 15).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Specify the number of contours or profiles in the **Levels** field. The maximum number of levels allowed is 100.

4. If you are generating a profile plot, enable the **Draw Profiles** option. In the resulting Profile Options Dialog Box (p. 2556) (Figure 31.14: The Profile Options Dialog Box (p. 1758)) you will define the profiles.

Figure 31.14: The Profile Options Dialog Box



   a. Set the "zero height" reference value for the profile (**Reference Value**) and the length scale factor for projection (**Scale Factor**). Any point on the profile with a value equal to the **Reference Value** will be plotted exactly on the defining surface. Values greater than the **Reference Value** will be projected ahead of the surface (in the direction of **Projection Dir.**) and scaled by **Scale Factor**), and values less than the **Reference Value** will be projected behind the surface and scaled.

      These parameters can be used to create fuller profiles when you need to display the variation in a variable that is small compared to the absolute value of the variable. Consider, for example, the display of temperature profiles when the temperature range in the domain is from 300 K to 310 K. The 10 K range in the temperature will be hard to detect when profiles are drawn using the default scaling (which will be based on the absolute magnitude of 310 K). To create a fuller profile, you can set the **Reference Value** to 300 and the profile **Scaling Factor** to 5 (for example) to magnify the display of the remaining 10 K range. In subsequent display of the profiles, the reference value of 300 will be effectively subtracted from the data before display so that the temperatures of 300 K will not be offset from the baselines. The profiles will then reflect only the variation of temperature from 300 K.

   b. Set the direction in which profiles are projected (**Projection Dir.**). In 2D, for example, a contour plot of pressure on the entire domain can be projected in the z direction to form a carpet plot, or a contour plot of y velocity on a sequence of y-coordinate slice lines can be projected in the y direction to form a series of velocity profiles (as shown in Figure 31.12: Profile Plot of y Velocity (p. 1756)).

   c. Click **Apply** and close the Profile Options Dialog Box (p. 2556).

5. Set any of the contour and profile plot options described below.

6. Click **Display** to draw the specified contours or profiles in the active graphics window.

The resulting display will include the specified number of contours or profiles of the selected variable, with the magnitude on each one determined by equally incrementing between the values shown in the **Min** and **Max** fields.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

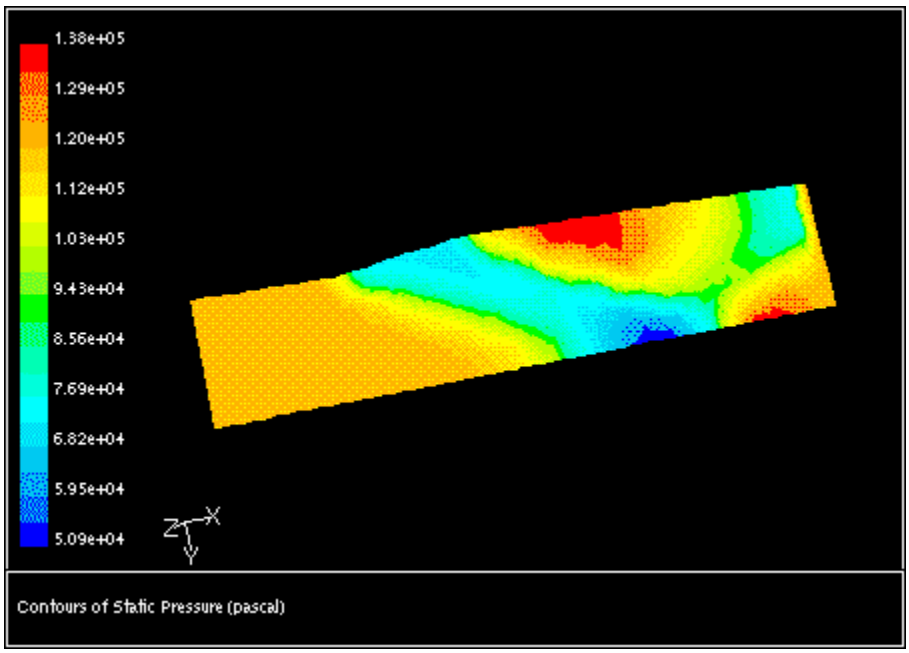
31.1.2.2. Contour and Profile Plot Options

The options mentioned in the procedure above include drawing color-filled contours/profiles (instead of the default line contours/profiles), specifying a range of values to be contoured or profiled, including portions of the mesh in the contour or profile display, choosing node or cell values for display, and storing the contour or profile plot settings.

31.1.2.2.1. Drawing Filled Contours or Profiles

Color-filled contour or profile plots show a contour or profile display containing a continuous color display (see Figure 31.15: Filled Contours of Static Pressure (p. 1759)), instead of just drawing lines representing specific values. Note that a color-filled profile display is often referred to as a "carpet plot". To generate a filled contour or profile plot, enable the **Filled** option in the Contours Dialog Box (p. 2676) during step 5 in the previous section.

Figure 31.15: Filled Contours of Static Pressure



To display smoothly shaded filled contours, you must enable lighting and select a lighting interpolation method other than **Flat** in the Display Options Dialog Box (p. 2573) or the Lights Dialog Box (p. 2588). You will not get smooth shading of filled contours if the **Clip to Range** option is turned on. Smooth shading of filled profiles is not available.

Note

The **Draw Profile** option settings are not saved with the contour plot definition. Once the dialog box is closed these options will revert to being disabled.

31.1.2.2.2. Specifying the Range of Magnitudes Displayed

By default, the minimum and maximum values contoured or profiled are set based on the range of values in the entire domain. This means that the color scale will start at the smallest value in the domain (shown in the **Min** field) and end at the largest value (shown in the **Max** field). If you are plotting contours or profiles on a subset of the domain (that is, on a surface), your plot may cover only the midrange of

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the color scale. For example, if blue corresponds to 0 and red corresponds to 10, and the values on your surface range only from 4 to 6, your plot will contain mostly green contours or profiles, since green is the color at the middle of the default color scale.

- To focus in on a smaller range of values, so that blue corresponds to 4 and red to 6, manually reset the range to be displayed. You can also use the minimum and maximum values on the selected surfaces—rather than in the entire domain—to determine the range. Another reason to manually set the range is if you are interested only in certain values. For example, if you want to determine the region where pressure exceeds a certain value, you can increase the minimum value for display so that the lower pressure values are not displayed.

- To manually set the contour/profile range, turn off the **Auto Range** option in the Contours Dialog Box (p. 2676). The **Min** and **Max** fields will become editable, and you can enter the new range of values to be displayed.

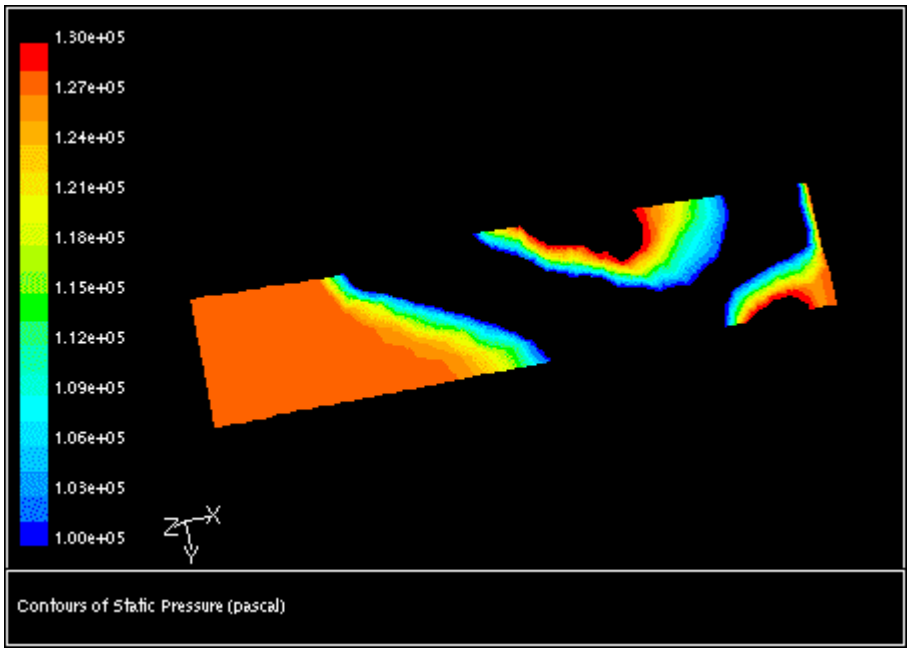
  – To show the default range at any time, click **Compute** and the **Min** and **Max** fields will be updated.

  – If you are drawing filled contours or profiles you can control whether or not values outside the prescribed **Min**/ **Max** range are displayed.

- To leave areas in which the value is outside the specified range empty (that is, draw no contours or profiles), enable the **Clip to Range** option. This is the default setting. If you turn **Clip to Range** off, values below the **Min** value will be colored with the lowest color on the color scale, and values above the **Max** value will be colored with the highest color on the color scale. Figure 31.16: Filled Contours with Clip to Range On (p. 1760) and Figure 31.17: Filled Contours with Clip to Range Off (p. 1761) show the results of enabling/disabling the **Clip to Range** option.

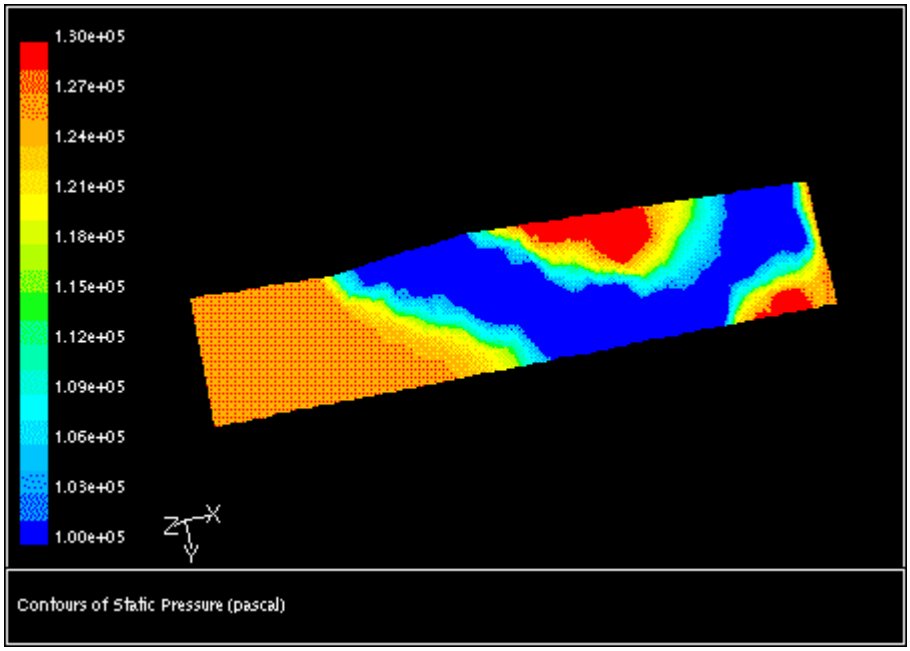
- To base the minimum and maximum values on the range of values on the selected surfaces, rather than in the entire domain, turn off the **Global Range** option in the Contours Dialog Box (p. 2676). The **Min** and **Max** values will be updated when you next click **Compute** or **Display**.

Figure 31.16: Filled Contours with Clip to Range On



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.17: Filled Contours with Clip to Range Off



31.1.2.2.3. Including the Mesh in the Contour Plot

For some problems, especially complex 3D geometries, you may want to include portions of the mesh in your contour or profile plot as spatial reference points. For example, you may want to show the location of an inlet and an outlet along with the contours. This is accomplished by turning on the **Draw Mesh** option in the Contours Dialog Box (p. 2676). The Mesh Display Dialog Box (p. 2163) will appear automatically when you enable the **Draw Mesh** option, and you can set the mesh display parameters there. When you click **Display** in the Contours Dialog Box (p. 2676), the mesh display, as defined in the Mesh Display Dialog Box (p. 2163), will be included in the contour or profile plot.

Note

The **Draw Mesh** option settings are not saved with the contour plot definition. Once the dialog box is closed these options will revert to being disabled.

To include the mesh with a contour plot that will be persisted with the case file, you can create a mesh plot definition and include both the mesh and contour plots in a scene. See Displaying a Scene (p. 1780) for more details.

31.1.2.2.4. Choosing Node or Cell Values

In ANSYS Fluent you can choose to display the computed cell-center values or values that have been interpolated to the nodes. By default, the **Node Values** option is turned on, and the interpolated values are displayed. For line contours or profiles, node values are always used. To display filled contours or profiles and to display the cell values, turn off the **Node Values** option. Filled contours/profiles of node values will show a smooth gradation of color, while filled contours/profiles of cell values may show sharp changes in color from one cell to the next.

For face-only functions (for example, **Wall Shear Stress**), the cell values that are displayed for boundary zone surfaces will actually be the face values. This is only true in the case of boundary zone surfaces created for postprocessing, where the actual cell values are used for the part of the surface that lies in

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the interior. These face values are more accurate, as face-only functions are computed on the faces and not on the cells. For more information about cell values, see Cell Values (p. 1931).

If you are plotting contours to show the effect of a porous medium or 2D fan boundary, to depict a shock wave, or to show any other discontinuities or jumps in the plotted variable, you should use cell values; if you use node values in such cases, the discontinuity will be smeared by the node averaging for graphics and will not be shown clearly in the plot.

31.1.2.2.5. Storing Contour Plot Settings

For frequently used combinations of contour variables and options, you can store the information needed to generate the contour plot as follows:

1. In the Contours Dialog Box (p. 2676), specify a **Setup** number.
2. Set up the desired information in the Contours Dialog Box (p. 2676).
3. Click **Display**.

   The settings for **Options**, **Contours of**, **Min**, **Max**, and **Surfaces** will be saved.

You can then change the **Setup** number to an unused value (that is, an ID for which no information has been saved) and generate a different contour plot.

To generate a plot using the saved setup information:

1. Change the **Setup** number back to the value for which you saved contour information.
2. Click **Display**.

You can save up to 10 different setups.

Important

Note that the number of contour **Levels**, the surfaces selected for display in the Mesh Display Dialog Box (p. 2163) (when the **Draw Mesh** option is activated), and the settings for profiles in the Profile Options Dialog Box (p. 2556) (when the **Draw Profiles** option is activated) will *not* be saved in the **Setup**, nor will the **Setup** be saved in the case file.

31.1.2.3. Creating and Using Contour Plot Definitions

You can create named contour plot definitions and save them for later use. This could be useful for demonstration purposes, scene creation, or for postprocessing the results of your simulation when varying the settings between runs (to name a few examples).

The number of contour plot definitions is not limited.

To create a contour plot definition:

Results → **Graphics** → **Contours** → **New...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the Contours Dialog Box (p. 2676), in the **Contour Name** text entry box, enter the name for your contour plot definition (or accept the default name `contour-`*`id`*).

   Important

   Contour and vector plot definitions must have unique names.

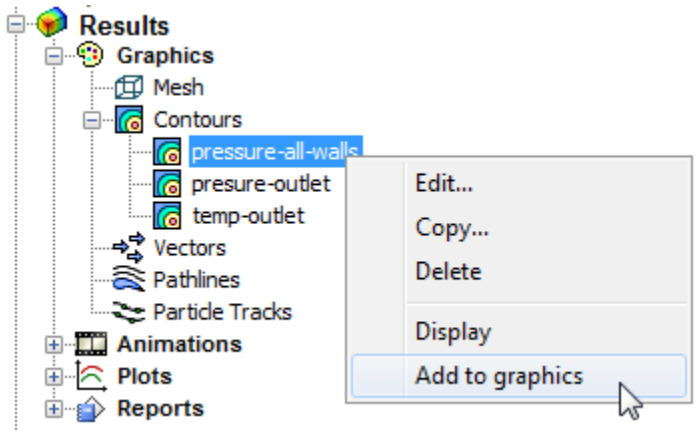
2. Set contour parameters and options.

   The **Draw Mesh** *and* **Draw Profile** *option settings are not saved with the contour plot definition. Once the dialog box is closed these options will revert to being disabled.*

3. Click **Save/Display**.

   The specified contour will be displayed in the active graphics window, and the new contour plot definition with the name you have entered will appear in the tree under **Contours**.

Once the contour plot definition is created, you can perform the following actions using the right-click menu commands:



- Edit the existing contour plot definition.
- Copy the contour plot definition.

  When creating a copy of the existing contour, you can modify contour settings in the **Contours** dialog box that will open.

- Delete the selected contour plot definition.
- Display the contour in the graphics window.
- Add the selected contour to the plot in the graphics window.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that while **Display** replaces the current contents of the graphics window, **Add to graphics** preserves the current graphics window contents.

Note

- You can move the colormap of contour plot definitions by left-clicking and dragging it to the desired location. The colormap can also be resized by dragging the corners of the box that appears when you hover over the colormap. *The colormap location is not saved with the contour plot definition.*
- For 2D cases, Fluent automatically creates a new surface when you click **Save/Display** in the **Contours** dialog box, if you have not selected any surfaces for display. New surfaces are only created for fluid zones that do not already have a surface defined.

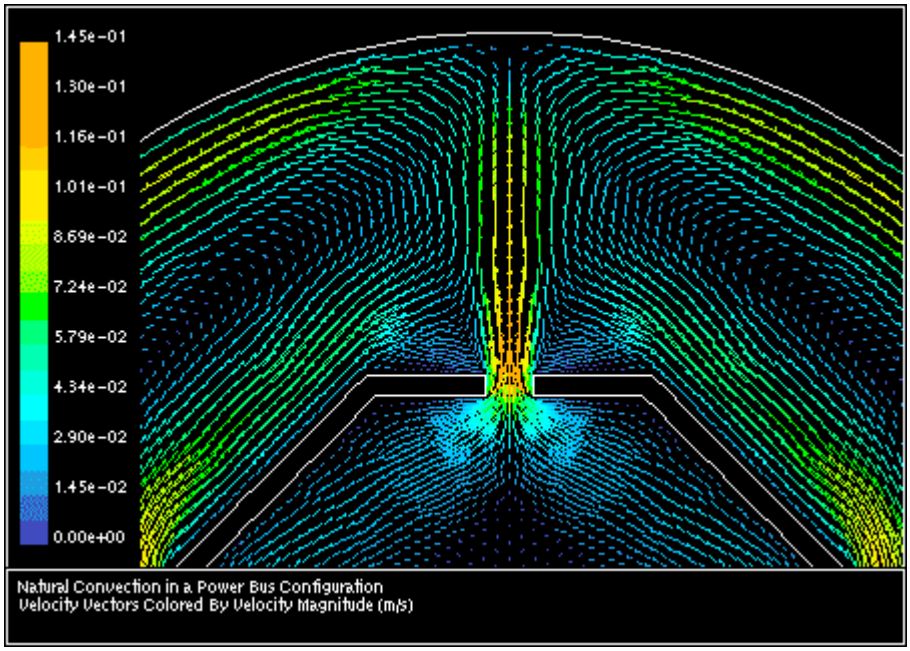
These commands are also available through the text user interface (`display/objects`).

31.1.3. Displaying Vectors

You can draw vectors in the entire domain, or on selected surfaces. By default, one vector is drawn at the center of each cell (or at the center of each facet of a data surface), with the length and color of the arrows representing the velocity magnitude (Figure 31.18: Velocity Vector Plot (p. 1764)). The spacing, size, and coloring of the arrows can be modified, along with several other vector plot settings. Velocity vectors are the default, but you can also plot vector quantities other than velocity. Note that cell-center values are always used for vector plots; you cannot plot node-averaged values.

For information about displaying vectors on a surface that sweeps through the domain, see Displaying Results on a Sweep Surface (p. 1782).

Figure 31.18: Velocity Vector Plot



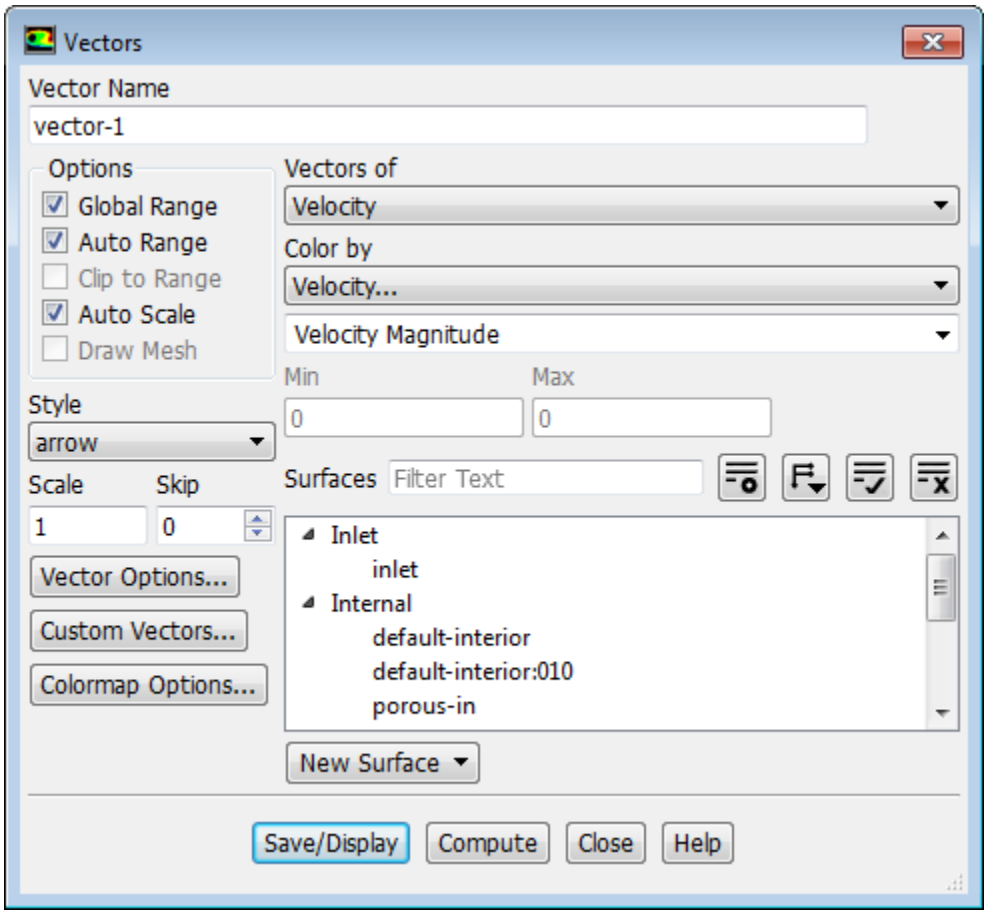
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.1.3.1. Generating Vector Plots

You can plot vectors using the Vectors Dialog Box (p. 2831) (Figure 31.19: The Vectors Dialog Box (p. 1765)).

Results → Graphics → Vectors **Edit...**

Figure 31.19: The Vectors Dialog Box



The procedure for generating a vector plot is as follows:

1. In the **Vectors of** drop-down list, select the vector quantity to be plotted. By default, only velocity and relative velocity are available, but you can create your own custom vectors as described in Creating and Managing Custom Vectors (p. 1769). Additional vector options are available for adjoint solver results, as described in Field Data (p. 2102).

2. In the **Surfaces** list, choose the surface(s) on which you want to display vectors. If you want to display vectors on the entire domain, select none of the surfaces in the list.

   If you want to select several surfaces of the same type, click and select **Surface Type** under **Group By**, which organizes the surfaces in a tree view that is grouped by surface type.

   Another shortcut is to use the **Filter Text** entry box to filter the **Surfaces** list to show only the surfaces that match the pattern you enter. For additional information on using the **Filter Text** entry box, see Filter Text Entry Boxes (p. 15).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Set any of the vector plot options as described in Vector Plot Options (p. 1766).

4. Click **Display** to draw the vectors in the active graphics window.

31.1.3.2. Displaying Relative Velocity Vectors

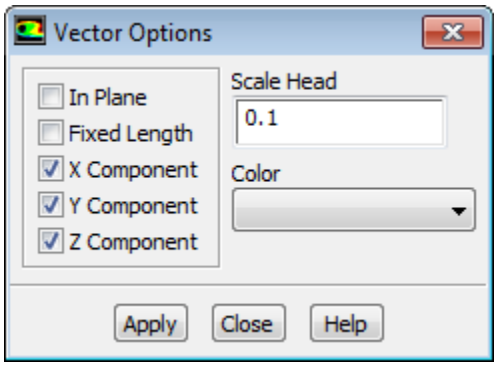
If you are solving your problem using one or more moving reference frames or moving meshes, you will have the option to display either the absolute vectors or the relative vectors. If you select **Velocity** (the default) in the **Vectors of** list, the vectors will be drawn based on the absolute, stationary reference frame. If you select **Relative Velocity**, the vectors will be drawn based on the reference frame of the **Reference Zone** in the Reference Values Task Page (p. 2502). See Setting the Reference Zone (p. 1927) for details. (If you are modeling a single moving reference frame, you need not specify the **Reference Zone**; the vectors will be drawn based on the moving reference frame.)

31.1.3.3. Vector Plot Options

The options mentioned in the procedure above include scaling the vector arrows, skipping the display of some vectors, displaying vectors in the plane of the data surface, displaying fixed-length or fixed-color vectors, displaying directional components of the vectors, specifying a range of values to be displayed, coloring the vectors by a different scalar field, including portions of the mesh in the vector display, and changing the style of the arrows or the scale of the arrowheads.

The most common options are set in the Vectors Dialog Box (p. 2831), and others are set in the Vector Options Dialog Box (p. 2557) (Figure 31.20: The Vector Options Dialog Box (p. 1766)), which you can open by clicking **Vector Options...** in the Vectors Dialog Box (p. 2831).

Figure 31.20: The Vector Options Dialog Box



31.1.3.3.1. Scaling the Vectors

By default, vectors are scaled automatically so that the arrows overlap minimally when no vectors are skipped. For instructions on thinning the vector display, see "Thinning" Pathlines (p. 1775). With the **Auto Scale** option, you can modify the **Scale** factor (which is set to 1 by default) to increase or decrease the vector scale from the default "auto scale". The main advantage of autoscaling is that the vector display with a scale factor of 1 will always be appropriate, regardless of the size of the domain, giving you a better starting point for fine-tuning the vector scale.

If you turn off the **Auto Scale** option, the vectors will be drawn at their actual sizes scaled by the scale factor (**Scale**, which is set to 1 by default). The "actual" size of a vector is the magnitude of the vector variable (velocity, by default) at the point where it is drawn. A vector drawn at a point where the velocity magnitude is 100 m/s is drawn 100 m long, whether the domain is 0.1 m or 1000 m. You can modify

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the vector scale by changing the value of **Scale** in the Vectors Dialog Box (p. 2831) until the size of the vectors (that is, the actual size multiplied by **Scale**) is satisfactory.

31.1.3.3.2. Skipping Vectors

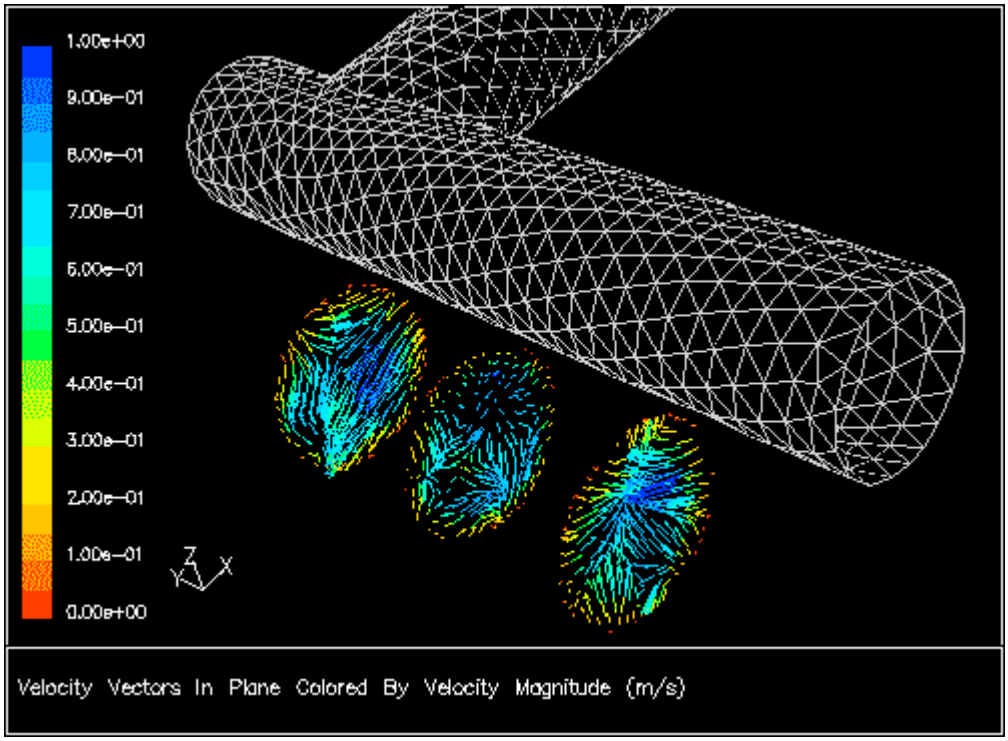
If your vector display is difficult to understand because there are too many arrows displayed, you can "thin out" the vectors by changing the **Skip** value in the Vectors Dialog Box (p. 2831). By default, **Skip** is set to 0, indicating that a vector will be drawn for each cell in the domain or for each face on the selected surface (for example, n vectors). If you increase **Skip** to 1, every other vector will be displayed, yielding $n/2$ vectors. If you increase **Skip** to 2, every third vector will be displayed, yielding $n/3$ vectors, and so on. The order of faces on the selected surface (or cells in the domain) will determine which vectors are skipped or drawn; therefore adaption will change the appearance of the vector display when a nonzero **Skip** value is used.

31.1.3.3.3. Drawing Vectors in the Plane of the Surface

For some problems, you may be interested in visualizing velocity (or other vector) components that are normal to the flow. These "secondary flow" components are usually much smaller than the components in the flow direction and are difficult to see when the flow direction components are also visible. To easily view the normal flow components, you can enable the **In Plane** option in the Vector Options Dialog Box (p. 2557). When this option is on, ANSYS Fluent will display only the vector components in the plane of the surface selected for display. If the selected surface is a cross-section of the flow domain, you will be displaying the components normal to the flow.

Figure 31.21: Velocity Vectors Generated Using the In Plane Option (p. 1767) shows velocity vectors generated using the **In Plane** option. Note that these vectors have been translated outside the domain, as described in Transforming Geometric Objects in a Scene (p. 1819), so that they can be seen more easily.

Figure 31.21: Velocity Vectors Generated Using the In Plane Option



31.1.3.3.4. Displaying Fixed-Length Vectors

By default, the length of a vector is proportional to its velocity magnitude. If you want all of the vectors to be displayed with the same length, you can enable the **Fixed Length** option in the Vector Options Dialog Box (p. 2557). To modify the vector length, adjust the value of the **Scale** factor in the Vectors Dialog Box (p. 2831).

31.1.3.3.5. Displaying Vector Components

All Cartesian components of the vectors are drawn by default, so that the arrow points along the resultant vector in physical space. However, sometimes one of the components, say, the x component, is relatively large. In such cases, you may want to suppress the x component and scale up the vectors, in order to visualize the smaller y and z components. To suppress one or more of the vector components, turn off the appropriate button(s) (**X**, **Y**, or **Z Component**) in the Vector Options Dialog Box (p. 2557).

31.1.3.3.6. Specifying the Range of Magnitudes Displayed

By default, the minimum and maximum vectors included in the vector display are set based on the range of vector-variable (velocity, by default) magnitudes in the entire domain. If you want to focus in on a smaller range of values, you can restrict the range to be displayed. The color scale for the vector display will change to reflect the new range of values. (You can also use the minimum and maximum values on the selected surfaces—rather than on the entire domain—to determine the range, or change the scalar field by which the vectors are colored from velocity magnitude to any other scalar, as described below.)

To manually set the range of velocity magnitudes (or the range of whatever scalar field is selected in the **Color by** drop-down list), turn off the **Auto Range** option in the Vectors Dialog Box (p. 2831). The **Min** and **Max** fields will become editable, and you can enter the new range of values to be displayed. For example, if you want to display velocity vectors only in regions where the velocity magnitude exceeds 150 m/s but is less than 300 m/s, you will change the value of **Min** to 150 and the value of **Max** to 300. Similarly, if you are coloring the vectors by static pressure, you can choose to display velocity vectors only in regions where the pressure is within a specified range. To show the default range at any time, click **Compute** and the **Min** and **Max** fields will be updated.

When you restrict the range of vectors displayed, you can also control whether or not values outside the prescribed **Min**/ **Max** range are displayed. To leave areas in which the value is outside the specified range empty (that is, draw no vectors), enable the **Clip to Range** option. This is the default setting. If you turn **Clip to Range** off, values below the **Min** value will be colored with the lowest color on the color scale, and values above the **Max** value will be colored with the highest color on the color scale. This feature is the same as the one available for displaying filled contours (see Figure 31.16: Filled Contours with Clip to Range On (p. 1760) and Figure 31.17: Filled Contours with Clip to Range Off (p. 1761)).

You can also choose to base the minimum and maximum values on the range of values on the selected surfaces, rather than the entire domain. To do this, turn off the **Global Range** option in the Vectors Dialog Box (p. 2831). The **Min** and **Max** values will be updated when you next click **Compute** or **Display**.

31.1.3.3.7. Changing the Scalar Field Used for Coloring the Vectors

If you want to color the vectors by a scalar field other than velocity magnitude (the default), you can select a different variable or function in the **Color by** drop-down list. Select the desired category in the upper list, and then choose one of the related quantities from the lower list. If you choose static pressure, for example, the length of the vectors will still correspond to the velocity magnitude, but the color of the vectors will correspond to the value of pressure at each point where a vector is drawn.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.1.3.3.8. Displaying Vectors Using a Single Color

If you want all of the vectors to be the same color, you can select the color to be used in the **Color** drop-down list in the Vector Options Dialog Box (p. 2557). If no color is selected (that is, if you choose the empty space at the top of the drop-down list—the default selection), the vector color will be determined by the **Color by** field specified in the Vectors Dialog Box (p. 2831). Single color vectors are useful in displays that overlay contours and vectors.

31.1.3.3.9. Including the Mesh in the Vector Plot

For some problems, especially complex 3D geometries, you may want to include portions of the mesh in your vector plot as spatial reference points. For example, you may want to show the location of an inlet and an outlet along with the vectors. This is accomplished by turning on the **Draw Mesh** option in the Vectors Dialog Box (p. 2831). The Mesh Display Dialog Box (p. 2163) will appear automatically when you enable the **Draw Mesh** option, and you can set the mesh display parameters there. When you click **Display** in the Vectors Dialog Box (p. 2831), the mesh display, as defined in the Mesh Display Dialog Box (p. 2163), will be included in the vector plot.

Note

The **Draw Mesh** option settings are not saved with the vector plot definition. Once the dialog box is closed these settings will revert to being disabled.

To include the mesh with a vector plot that will be persisted with the case file, you can create a mesh plot definition and include both the mesh and vector plots in a scene. See Displaying a Scene (p. 1780) for more details.

31.1.3.3.10. Changing the Arrow Characteristics

There are five different styles available for drawing the vector arrows. Choose **cone**, **filled-arrow**, **arrow**, **harpoon**, or **headless** in the **Style** drop-down list in the Vectors Dialog Box (p. 2831). The default arrow style is **harpoon**.

If you choose a vector arrow style that includes heads, you can control the size of the arrowhead by modifying the **Scale Head** value in the Vector Options Dialog Box (p. 2557).

31.1.3.4. Creating and Managing Custom Vectors

In addition to the velocity vector quantity provided by ANSYS Fluent, you can also define your own custom vectors to be plotted. This capability is available with the Custom Vectors Dialog Box (p. 2558).

Any custom vectors that you define will be saved in the case file the next time that you save it. You can also save your custom vectors to a separate file, so that they can be used with a different case file.

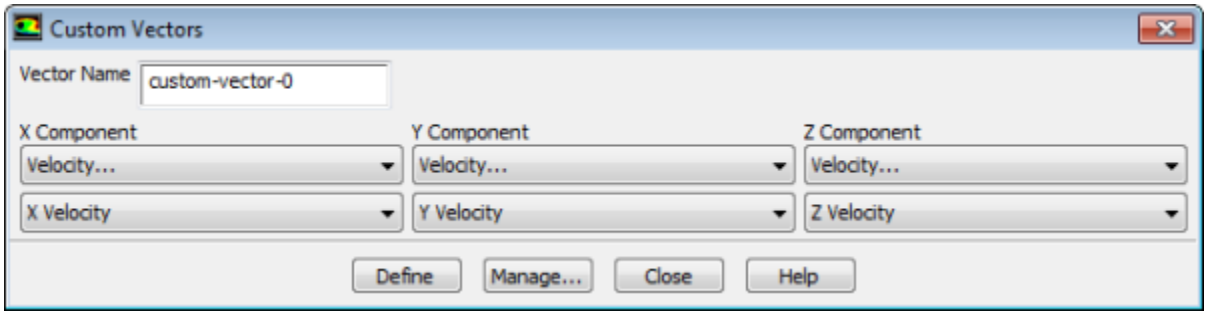
31.1.3.4.1. Creating Custom Vectors

To create your own custom vector, you will use the Custom Vectors Dialog Box (p. 2558) (Figure 31.22: The Custom Vectors Dialog Box (p. 1770)). This dialog box enables you to define custom vectors based on existing quantities. Any vectors that you define will be added to the **Vectors of** list in the Vectors Dialog Box (p. 2831).

To open the Custom Vectors Dialog Box (p. 2558), click **Custom Vectors...** in the Vectors Dialog Box (p. 2831).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.22: The Custom Vectors Dialog Box



The steps for creating a custom vector are as follows:

1. Specify the name of the custom vector in the **Vector Name** field.

   Important

   Do not specify a name that is already used for a standard vector (for example, `velocity` or `relative-velocity`).

2. Select the variable or function for the *x* component of the vector in the **X Component** drop-down list. First select the desired category in the upper list; you may then select a related quantity in the lower list. For an explanation of the variables in the list, see Field Function Definitions (p. 1931).

3. Repeat the step above to select the variable or function for the y component (and, in 3D, the z component) of the custom vector.

   Important

   You can use the **Custom Vectors** option to plot vectors in solid cell zones. The scalars that are selected in the *x*, *y* (and, in 3D, the *z*) components, and that are valid in solid regions, will have vector plots displayed in the solid cell zones. Note that if a vector has no valid components in the solid region, then that vector will not be plotted in the solid region. However, if at least one component of the vector is valid in the solid region, then only that component of the vector will be plotted.

4. Click **Define**.

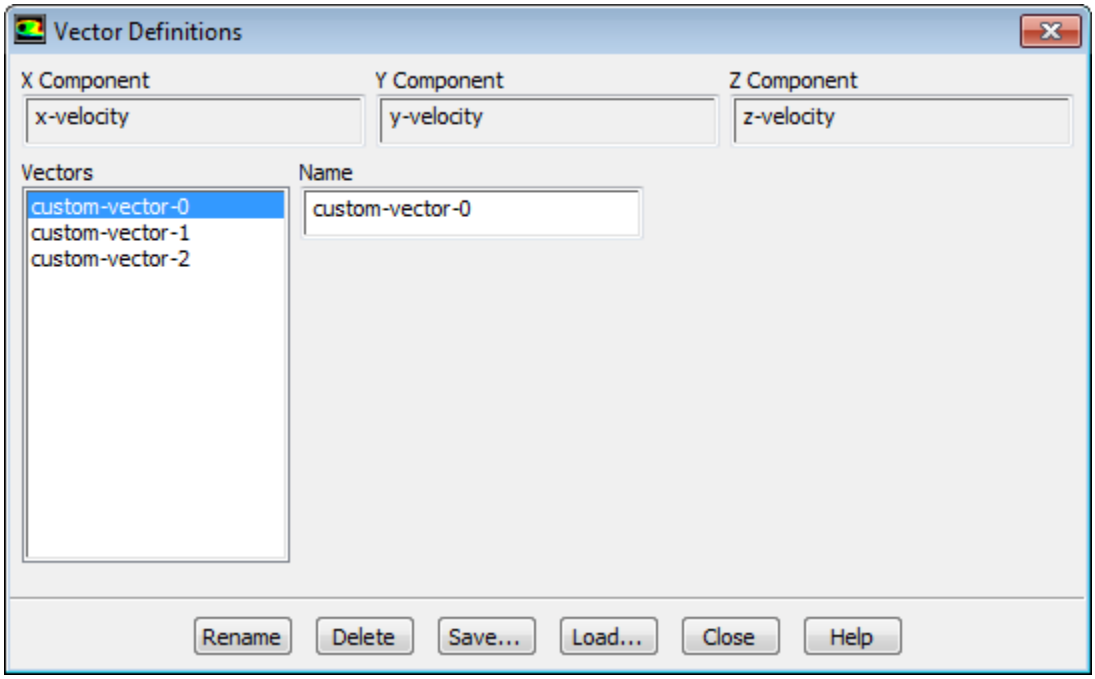
31.1.3.4.2. Manipulating, Saving, and Loading Custom Vectors

Once you have defined your vectors, you can manipulate them using the Vector Definitions Dialog Box (p. 2558) (Figure 31.23: The Vector Definitions Dialog Box (p. 1771)). You can display a vector definition to be sure that it is correct, delete the vector if you decide that it is incorrect and must be redefined, or give the vector a new name. You can also save custom vectors to a file or read them from a file. The custom vector file enables you to transfer custom vectors between case files.

To open the Vector Definitions Dialog Box (p. 2558), click **Manage...** in the Custom Vectors Dialog Box (p. 2558).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.23: The Vector Definitions Dialog Box



The following actions can be performed in the Vector Definitions Dialog Box (p. 2558):

- To check the definition of a vector, select it in the **Vectors** list. Its definition will be displayed in the **X Component**, **Y Component**, and **Z Component** fields at the top of the dialog box. This display is for informational purposes only; you cannot edit it. If you want to change a vector definition, you must delete the vector and define it again in the Custom Vectors Dialog Box (p. 2558).
- To delete a vector, select it in the **Vectors** list and click **Delete**.
- To rename a vector, select it in the **Vectors** list, enter a new name in the **Name** field, and click **Rename**.

Important

Do not specify a name that is already used for a standard vector (for example, `velocity` or `relative-velocity`).

- To save all the vectors in the **Vectors** list to a file, click **Save...** and specify the file name in The Select File Dialog Box (p. 19).
- To read custom vectors from a file that you saved as described above, click **Load** and specify the file name in The Select File Dialog Box (p. 19). (Custom vectors are valid Scheme functions, and can also be loaded with the **File/Read/Scheme...** ribbon tab item, as described in Reading Scheme Source Files (p. 57).)

31.1.3.5. Creating and Using Vector Plot Definitions

You can create named vector plot definitions and save them for later use. See Creating and Using Contour Plot Definitions (p. 1762) for additional information on plot definitions.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The steps for creating and using vector plot definitions are similar to the steps for Generating Vector Plots (p. 1765), with the addition of a field for naming the vector plot definition.

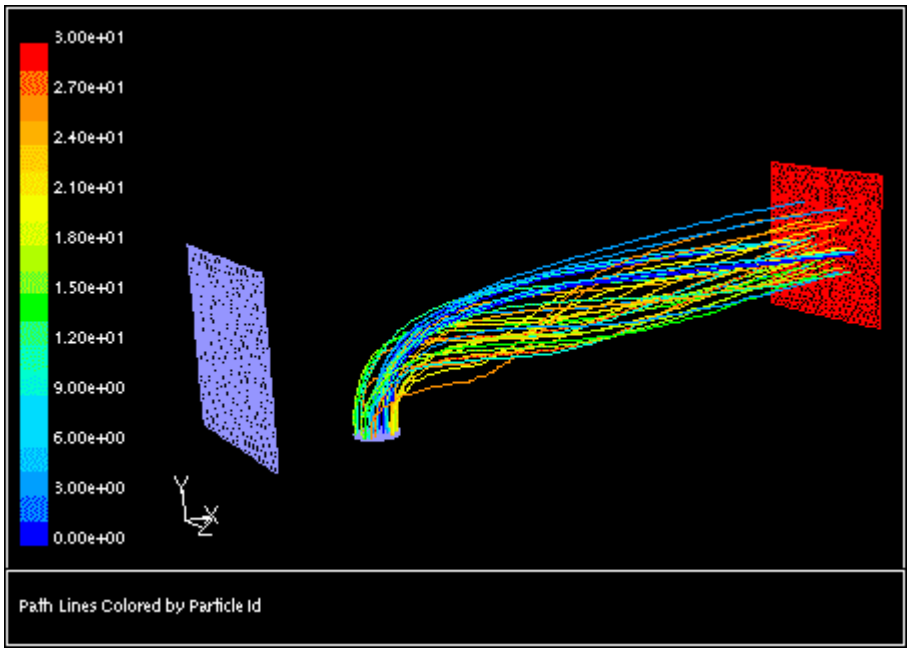
Note

You can move the colormap of a vector plot definition by left-clicking and dragging it to the desired location. The colormap can also be resized by dragging the corners of the box that appears when you hover over the colormap. *The colormap location is not saved with the vector plot definition.*

31.1.4. Displaying Pathlines

Pathlines are used to visualize the flow of massless particles in the problem domain. The particles are released from one or more surfaces that you have created by clicking **Create** in the **Surface** group box of the **Setting Up Domain** ribbon tab (see Creating Surfaces and Cell Registers for Displaying and Reporting Data (p. 1709)). A **line** or **rake** surface (see Line and Rake Surfaces (p. 1718)) is most commonly used. Figure 31.24: Pathline Plot (p. 1772) shows a sample plot of pathlines.

Figure 31.24: Pathline Plot



Note that the display of discrete-phase particle trajectories is discussed in Displaying of Trajectories (p. 1330).

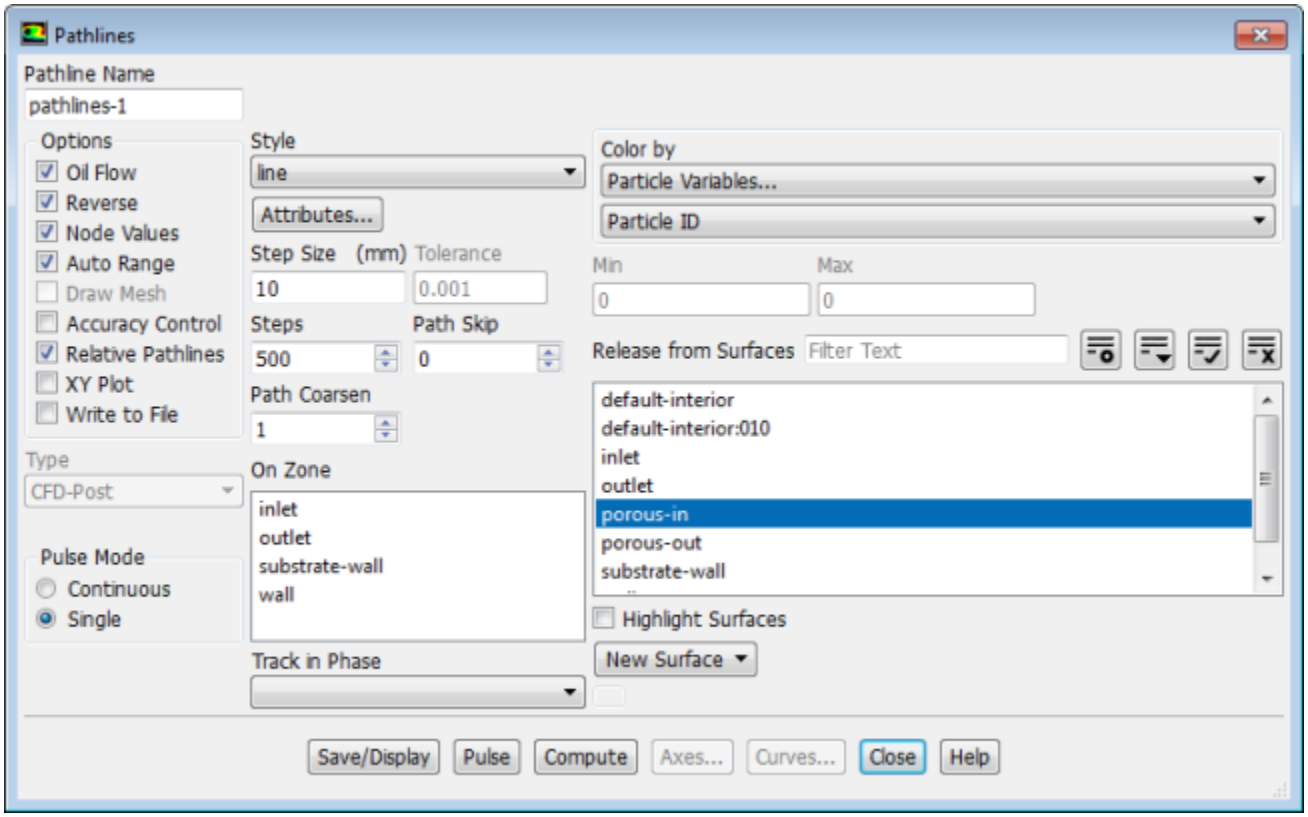
31.1.4.1. Steps for Generating Pathlines

You can plot pathlines using the Pathlines Dialog Box (p. 2771) (Figure 31.25: The Pathlines Dialog Box (p. 1773)).

Results → Graphics → Pathlines → Edit...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.25: The Pathlines Dialog Box



The basic steps for generating pathlines are as follows:

1. Select the surface(s) from which to release the particles in the **Release From Surfaces** list.

2. (Eulerian Multiphase model only) From the **Track in Phase** drop-down list, select the phase in which pathlines will be tracked. If you want to track pathlines in the mixture of all phases, select **all-phase**. By default, particles are tracked in the primary phase.

3. Set the step size and the maximum number of steps. The **Step Size** sets the length interval used for computing the next position of a particle. Note that particle positions are always computed when particles enter/leave a cell; even if you specify a very large step size, the particle positions at the entry/exit of each cell will still be computed and displayed. The value of **Steps** sets the maximum number of steps a particle can advance. A particle will stop when it has traveled this number of steps or when it leaves the domain. One simple rule of thumb to follow when setting these two parameters is that if you want the particles to advance through a domain of length L, the **Step Size** times the number of **Steps** should be approximately equal to L.

4. Set any of the pathline plot options described in Options for Pathline Plots (p. 1773).

5. Click **Display** to draw the pathlines, or click **Pulse** to animate the particle positions. The **Pulse** button will become the **Stop !** button during the animation, and you must click **Stop !** to stop the pulsing.

31.1.4.2. Options for Pathline Plots

You can include the mesh in the pathline display, control the style of the pathlines (including the twisting of ribbon-style pathlines), and color them by different scalar fields and control the color scale. You can also "thin" the pathline display, trace the particle positions in reverse, and draw "oil-flow" pathlines. If you are "pulsing" the pathlines, you can control the pulse mode. If you are using larger

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

time step size for calculations then you can control the accuracy of the pathline by specifying tolerance. In addition to the regular pathline display, you can also generate an XY plot of a specified quantity along the pathline trajectories. Finally, you can choose node or cell values for display (or plotting).

31.1.4.2.1. Including the Mesh in the Pathline Display

For some problems, especially complex 3D geometries, you may want to include portions of the mesh in your pathline display as spatial reference points. For example, you may want to show the location of an inlet and an outlet along with the pathlines (as in Figure 31.24: Pathline Plot (p. 1772)). This is accomplished by turning on the **Draw Mesh** option in the Pathlines Dialog Box (p. 2771). The Mesh Display Dialog Box (p. 2163) will appear when you enable the **Draw Mesh** option, where you can set the mesh display parameters. When you click **Display** in the Pathlines Dialog Box (p. 2771), the mesh display, as defined in the Mesh Display Dialog Box (p. 2163), will be included in the plot of pathlines.

Note

The **Draw Mesh** option settings are not saved with the pathline definition. Once the dialog box is closed these settings will revert to being disabled. If you want these settings persisted within the current session, you can use the non-persistent **Pathlines** dialog box.

Postprocessing → Graphics → Pathlines → Edit...

31.1.4.2.2. Controlling the Pathline Style

Pathlines can be displayed as lines (with or without arrows), ribbons, cylinders (coarse, medium, or fine), triangles, spheres, or a set of points. You can choose **line**, **line-arrows**, **point**, **sphere**, **ribbon**, **triangle**, **coarse-cylinder**, **medium-cylinder**, or **fine-cylinder** in the **Style** drop-down list in the Pathlines Dialog Box (p. 2771). Pulsing can be done only on **point**, **sphere**, or **line** styles.

Once you have selected the pathline style, click **Style Attributes...** to set the pathline thickness and other parameters related to the selected **Style**:

- If you are using the **line** or **line-arrows** style, set the **Line Width** in the Path Style Attributes Dialog Box (p. 2559) that appears when you click **Style Attributes...**. For **line-arrows** you will also set the **Spacing Factor**, which controls the spacing between the lines. The size of the arrow heads can be adjusted by entering a value in the **Scale** text-entry box.

- If you are using the **point** style, you will set the **Marker Size** in the Path Style Attributes Dialog Box (p. 2559). The thickness of the pathline will be the thickness of the marker.

- If you are using the **sphere** style, you will set the **Diameter** and the **Detail** in the Path Style Attributes Dialog Box (p. 2559).

  The best diameter to use will depend on the dimensions of the domain, the view, and the particle density. However, an adequate starting point would be a diameter on the order of 1/4 of the average cell size or 1/4 step size. Units for the **Diameter** field correspond to the mesh dimensional units.

  The level of detail applied to the graphical rendering of the spheres can be controlled using the **Detail** field in the Path Style Attributes Dialog Box (p. 2559). The level of detail uses integer values ranging from 4 to 50. Note that the performance of the graphical rendering will be better when using a small level of detail, that is, very coarse spheres, such as 6 or 8. The rendering performance significantly decreases with higher levels of detail. You should gradually increase the detail to determine the best-case scenario between performance and quality.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Also note that to take full advantage of spherical rendering, lighting should be turned on in the view. The Gouraud setting provides much smoother looking spheres than the Flat setting and better performance than the Phong setting. For more information on lighting, see Adding Lights (p. 1796).

- If you are using the **triangle** or any of the **cylinder** styles, you will set the **Width** in the Path Style Attributes Dialog Box (p. 2559). For triangles, the specified value will be half the width of the triangle's base, and for cylinders, the value will be the cylinder's radius.

- If you are using the **ribbon** style, clicking **Style Attributes...** opens the Ribbon Attributes Dialog Box (p. 2560), in which you can set the ribbon's **Width**. You can also specify parameters for twisting the ribbon pathlines. In the **Twist By** drop-down list, you can select a scalar field on which the pathline twisting is based (for example, helicity). Select the desired category in the upper list and then select a related quantity in the lower list. The twisting may not be displayed smoothly because the scalar field by which you are twisting the pathline is calculated at cell centers only (and not interpolated to a particle's position). The **Twist Scale** sets the amount of twist for the selected scalar field. To magnify the twist for a field with very little change, increase this factor; to display less twist for a field with dramatic changes, decrease this factor.

  When you click **Compute**, the **Min** and **Max** fields will be updated to show the range of the **Twist By** scalar field.

31.1.4.2.3. Controlling Pathline Colors

By default, the pathlines are colored by the particle ID number. That is, each particle's path will be a different color. You can also choose the color based on the surface from where the pathlines were released from using the surface ID as the particle variable. You can choose to color the pathlines by any of the scalar fields in the **Color by** drop-down list. Select the desired category in the upper list and then select a related quantity in the lower list. If you color the pathlines by velocity magnitude, for example, each particle's path will be colored depending on the speed of the particle at each point in the path.

The range of values of the selected scalar field will, by default, be the upper and lower limits of that field in the entire domain. The color scale will map to these values accordingly. If you prefer to restrict the range of the scalar field, turn off the **Auto Range** option (under **Options**) and set the **Min** and **Max** values manually beneath the **Color by** list. If you color the pathlines by velocity, and you limit the range to values between 30 and 60 m/s, for example, the "lowest" color will be used when the particle speed falls below 30 m/s and the "highest" color will be used when the particle speed exceeds 60 m/s. To show the default range at any time, click **Compute** and the **Min** and **Max** fields will be updated.

31.1.4.2.4. "Thinning" Pathlines

If your pathline plot is difficult to understand because there are too many paths displayed, you can "thin out" the pathlines by changing the **Path Skip** value in the Pathlines Dialog Box (p. 2771). By default, **Path Skip** is set to 0, indicating that a pathline will be drawn from each face on the selected surface (for example, n pathlines). If you increase **Path Skip** to 1, every other pathline will be displayed, yielding $n/2$ pathlines. If you increase **Path Skip** to 2, every third pathline will be displayed, yielding $n/3$, and so on. The order of faces on the selected surface will determine which pathlines are skipped or drawn; therefore adaption will change the appearance of the pathline display when a nonzero **Path Skip** value is used.

31.1.4.2.5. Coarsening Pathlines

To further simplify pathline plots, and reduce plotting time, a coarsening factor can be used to reduce the number of points that are plotted. Providing a coarsening factor of n, will result in each n^{th} point being plotted for a given pathline in any cell. This coarsening factor is specified in the Pathlines Dialog

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Box (p. 2771), in the **Path Coarsen** field. For example, if the coarsening factor is set to 2, then ANSYS Fluent will plot alternate points.

Important

Note that if any particle or pathline enters a new cell, this point will always be plotted.

31.1.4.2.6. Reversing the Pathlines

If you are interested in determining the source of a particle for which you know the final destination (for example, a particle that leaves the domain through an exit boundary), you can reverse the pathlines and follow them from their destination back to their source. To do this, enable the **Reverse** option in the Pathlines Dialog Box (p. 2771). All other inputs for defining the pathlines will be exactly the same as for forward pathlines; the only difference is that the surface(s) selected in the **Release From Surfaces** list will be the final destination of the particles instead of their source.

31.1.4.2.7. Plotting Oil-Flow Pathlines

If you want to display "oil-flow" pathlines (that is, pathlines that are constrained to lie on a particular boundary), enable the **Oil Flow** option in the Pathlines Dialog Box (p. 2771). You will then need to select a single boundary zone in the **On Zone** list. The selected zone is the boundary on which the oil-flow pathlines will lie.

31.1.4.2.8. Controlling the Pulse Mode

If you are going to use the **Pulse** button in the Pathlines Dialog Box (p. 2771) to animate the pathlines, you can choose one of two pulse modes for the release of particles that follow the pathlines. To release a single wave of particles, select the **Single** option under **Pulse Mode**. To release particles continuously from the initial positions, select the **Continuous** option.

31.1.4.2.9. Controlling the Accuracy

If you are using large time step size for the calculation, there might be significant error introduced while calculating the pathlines. To control this error, select **Accuracy Control** and specify the value of **Tolerance**. The tolerance value will be taken in to consideration while calculating the pathlines for each time step.

31.1.4.2.10. Plotting Relative Pathlines

If you want to display the pathlines relative to the moving reference frame, enable the **Relative Pathlines** option in the Pathlines Dialog Box (p. 2771). You will then need to select the surfaces from the **Release from Surfaces** list.

31.1.4.2.11. Generating an XY Plot Along Pathline Trajectories

If you want to generate an XY plot along the trajectories of the pathlines you have defined, enable the **XY Plot** option in the Pathlines Dialog Box (p. 2771). The **Color by** drop-down list will be replaced by **Y Axis Function** and **X Axis Function** lists. Select the variable to be plotted on the y-axis in the **Y Axis Function** list, and specify whether you want to plot this quantity as a function of the **Time** elapsed along the trajectory, or the **Path Length** along the trajectory by selecting the appropriate item in the **X Axis Function** drop-down list. Specify the **Step Size**, number of **Steps**, and other parameters as usual for a standard pathline display. Then click **Plot** to display the XY plot.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Once you have generated an XY plot, you may want to save the plot data to a file. You can read this file into ANSYS Fluent at a later time and plot it alone using the File XY Plot Dialog Box (p. 2599), as described in XY Plots of File Data (p. 1844), or add it to a plot of solution data, as described in Including External Data in the Solution XY Plot (p. 1844).

To save the plot data to a file, enable the **Write to File** option in the Pathlines Dialog Box (p. 2771). The **Plot** button changes to a **Write...** button. Clicking **Write...** opens The Select File Dialog Box (p. 19), in which you can specify a name and save a file containing the plot data. The format of this file is described in XY Plot File Format (p. 1850).

31.1.4.2.12. Saving Pathline Data

To save pathline data to a file, perform the following steps:

1. Enable the **Write to File** option in the Pathlines Dialog Box (p. 2771) (Figure 31.25: The Pathlines Dialog Box (p. 1773)).

2. In the **Type** drop-down list, select one of the following types of files:

   - **Standard** for `FIELDVIEW` (`.fvp`) format
   - **Geometry** for `.ibl` format (which can be read by GAMBIT)
   - **EnSight** format

   Important

   If you plan to write the pathline data in **EnSight** format, you should first verify that you have already written the files associated with the **EnSight Case Gold** file type by using the **File/Export...** ribbon tab option (see EnSight Case Gold Files (p. 76)).

   For further information about the files that are written for any of these types, refer to the appropriate section following these steps.

3. Choose to color the pathlines by any of the scalar fields in the **Color by** drop-down lists.

4. Select the surface(s) from which to release the particles in the **Release From Surfaces** list.

5. If you selected **EnSight** under **Type**, you will need to specify the **EnSight Encas File Name**. Use **Browse...** to select the `.encas` file that was created when you exported the file with the **File/Export...** ribbon tab option. If you do not make a selection, then you will need to create an appropriate `.encas` file manually.

   You can also select the number of **Time Steps For EnSight Export**. This number directly determines how many time levels will be available for animation in EnSight.

6. Click **Write...** to open The Select File Dialog Box (p. 19), in which you can specify a name and save a file containing the pathline data.

To initiate saving pathline data through the text command interface enter the following TUI command:

```
display/path-lines/write-to-files
```

In addition to pathline data, you can also export particle data in either **Standard**, **EnSight** or **Geometry** type. For information on exporting particle data in **FIELDVIEW** (standard), **EnSight** or `.ibl` (geometry) format, refer to Exporting Steady-State Particle History Data (p. 84).

31.1.4.2.12.1. Standard Type

If **Standard** is selected under **Type**, ANSYS Fluent will write the file in **FIELDVIEW** format, which can be exported and read into **FIELDVIEW**. The **FIELDVIEW** ASCII Particle Path Format is licensed from Intelligent Light, proprietor of an independent visualization software package (http://www.ilight.com). The file name that you use for saving the data must have a `.fvp` extension. You also have the ability to retrieve and display the particle and pathline trajectories from the file.

If the case is steady-state, the particle path information will be written in ASCII format. For transient or unsteady-state cases, the BINARY format must be used. The **FIELDVIEW** file contains a set of paths, where each path consists of a series of points. At every point the spatial location and selected variables are defined. A full description of the ASCII and BINARY formats can be found in Appendix K - Particle Path Formats of FIELDVIEW's Reference Manual [2] (p. 2887), available to licensed **FIELDVIEW** users.

The following is an example of the **FIELDVIEW** format for a steady-state case:

```
FVPARTICLES 2 1
Tag Names
0
Variable Names
2
time
particle_id
3
0.2 0.8 1.3 0.2 0
0.3 0.9 1.3 0.4 0
0.5 1.1 1.3 0.6 0
```

The beginning of the file displays header information. `Tag Names` cannot be specified when the file is exported from ANSYS Fluent, and hence will always be `0`. ANSYS Fluent enables you to export two variables, which are listed under `Variable Names`: the first is determined by the scalar fields selected in the **Color by** drop-down lists (`time` in the example above); the second is always `particle_id`.

The rest of the file contains information about each path. A path section begins by listing the total number of points for the path. Then a line of data is presented for each point, with the X, Y, and Z locations listed in the first three columns and the variable information in the fourth and fifth columns. The example above presents a single pathline consisting of three points; the time ranges from `.2` to `.6`, and the ID of the particle is `0`.

31.1.4.2.12.2. Geometry Type

If **Geometry** is selected under **Type**, the file will be written in `.ibl` format. The resulting file contains particle paths in the form of a curve that can be read in GAMBIT. The following is an example of a **Geometry** file format that contains multiple curves:

```
Closed Index Arclength
Begin section ! 1
 Begin curve ! 1
 1                  185.61      0                 23.26
   2       88.90000000000001      0                -89.67

   Begin curve ! 2
   1       88.89999999999569      0      -89.6699999999997
   2       76.90221619148909      0     -101.2290490001453
   3       62.92208239159677      0     -110.2907424975297
   4       47.47166726362848      0     -116.5231659809653
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
5       31.11689338997181      0     -119.6980363161113
6       14.45680848476821      0     -119.6990633707006
7      -1.898356710978934      0     -116.5262095254603
8      -17.34954014966171      0     -110.2956910520416
9      -31.33079110697006      0     -101.2357213074894
10       -43.330000000007      0     -89.67815166483965

Begin curve ! 3
1                  -43.33      0     -89.67815166485001
2                 -175.56      0         64.69066040289
```

The above example demonstrates how multiple curves can be imported; single curves may also be imported. After importing this file into GAMBIT, the file is read by first looking for a `Begin curve` string and then looking for the X, Y, and Z coordinates under the `Begin curve` line.

31.1.4.2.12.3. EnSight Type

By selecting **EnSight** under **Type**, you can generate files with the following extensions:

- `.mpg`
- `.mscl`
- `.encas`

An `.mpg` file will be written for every time step specified in the **Time Steps For EnSight Export** field. A sequential number will be appended to the `.mpg` extension to indicate the time step. Each file contains a header that lists the time at which the data was exported, as well as three columns listing the X, Y, and Z coordinates for every particle at that particular time step.

The following is an example of a file called `particle.mpg0003`, which contains data for nine particles at the third time step:

```
File is written from fluent in ensight measured particle format for
t = 2.42813e-04
particle coordinates
  9
  1-7.27734e-05 1.91710e-03 4.69093e-03
  2-1.75772e-04 1.97040e-03 3.92842e-03
  3-2.26051e-04 2.10134e-03 5.63228e-03
  4-1.16390e-04 2.32442e-03 5.23423e-03
  5-6.32735e-04 2.53326e-03 5.70791e-03
  6-9.69431e-04 2.37006e-03 5.27602e-03
  7-6.77868e-04 2.92054e-03 4.11570e-03
  8-9.78029e-04 2.75717e-03 4.13314e-03
  9-8.54859e-04 3.73727e-03 2.23796e-03
```

An `.mscl` file will be written for every time step specified in the **Time Steps For EnSight Export** field. A sequential number will be appended to the `.mscl` extension to indicate the time step. Each file contains the scalar information (specified under **Color By**) for every particle at a particular time step.

The following is an example of a file called `particle.mscl0006`, which captures **Particle ID** data for nine particles at the sixth time step:

```
particle id
 0.00000e+00 6.00000e+00 1.20000e+01 1.80000e+01 2.40000e+01 3.00000e+01
 3.60000e+01 4.20000e+01 4.80000e+01
```

A new `.encas` file will be written if a selection is made under **EnSight Encas File Name**. This new file is a modified version of the `.encas` file selected with the **Browse...** button, and contains information about all of the related files (including geometry, velocity, scalar and coordinate files). The name of the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

new file will be the root of the original file with `.new` appended to it (for example if `test.encas` is selected, a file named `test.new.encas` will be written). It is this new file that should be read into **EnSight**.

The following is an example of a file called `spray2-unsteady.new.encas`, that refers to the files generated when the data was originally exported as an **EnSight Case Gold** file type (`.geo`, `.vel`, `.sc11`, and `.sc12`) and the files created during the pathline data export (`.mpg` and `.mscl`):

```
FORMAT
type: ensight gold
GEOMETRY
model: spray2-unsteady.geo
measured: 1 particle.mpg****
VARIABLE
scalar per measured node: 1 particle-id particle.mscl****
scalar per node: pressure                     spray2-unsteady.scl1
scalar per node: pressure-coefficient         spray2-unsteady.scl2
vector per node: velocity                     spray2-unsteady.vel
TIME
time set: 1 Model
number of steps: 10
filename start number:   1
filename increment:      1
time values: 0.00000e+00 1.21406e-04 2.42813e-04 3.64219e-04 4.85626e-04
6.07032e-04 7.28438e-04 8.49845e-04 9.71251e-04 1.09266e-03
```

31.1.4.2.13. Choosing Node or Cell Values

In ANSYS Fluent you can determine the scalar field value at a particle location using the computed cell-center values or values that have been interpolated to the nodes. By default, the **Node Values** option is turned on, and the interpolated values are used. If you prefer to use the cell values, turn the **Node Values** option off. Note that for face-only functions like **Wall Shear Stress**, the cell value is the area-weighted average from the face values that define that cell as `c0`.

If you are plotting pathlines to show the effect of a porous medium or 2D fan boundary, to depict a shock wave, or to show any other discontinuities or jumps in the plotted variable, you should use cell values; if you use node values in such cases, the discontinuity will be smeared by the node averaging for graphics and will not be shown clearly in the plot.

31.1.4.3. Creating and Using Pathline Definitions

You can create named pathline definitions and save them for later use. See Creating and Using Contour Plot Definitions (p. 1762) for additional information on graphics object definitions.

The steps for creating and using pathline definitions are similar to the Steps for Generating Pathlines (p. 1772), with the addition of a field for naming the pathline definition.

Pathline definitions can be included in scenes as long as the **XY Plot** option is not enabled.

31.1.5. Displaying a Scene

Scenes can be used to display multiple graphics plots within a single window. For example, you could overlay contours of pressure across a valve with velocity vectors and the mesh at the same location. Scenes allow you to modify the transparency of each plot so that you can emphasize a particular plot or view.

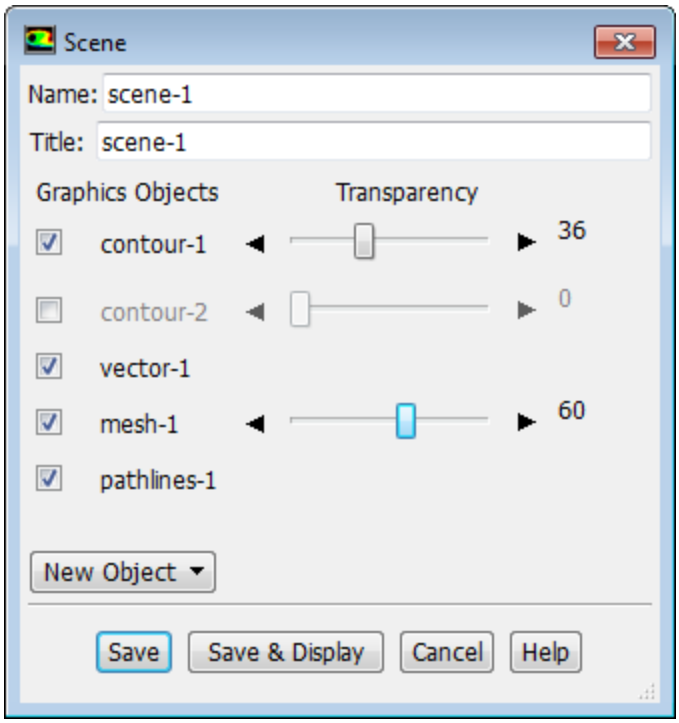
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.1.5.1. Generating a Scene

You can plot a scene made up of multiple graphics objects using the Figure 31.26: The Scene Dialog Box (p. 1781).

Results → Scene New...

Figure 31.26: The Scene Dialog Box



The basic steps for creating a scene:

1. (Optional) Enter a **Name** for the scene.
2. (Optional) Enter a **Title** for the scene, which will appear in the graphics window tab where the scene is displayed.
3. Select the **Graphics Objects** that you want to include in the scene. *If you want to include additional mesh, contour, vector, or pathline plots in the scene, create them from the **New Object** drop-down list.*

   *Note that pathlines with the **XY Plot** option enabled are not compatible with scenes.*
4. (Optional) You can modify the transparency of the selected graphics objects using the **Transparency** slider. *If the slider is completely to the right the object will be fully transparent.*
5. Click **Save** or **Save & Display**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scene Colormaps

Once the scene is displayed, you can move the colormaps by left-click and drag. Colormaps can also be resized by dragging the corners of the box that appears when you hover over the colormap.

Tip

You can specify a different colormap scale for each graphic object to make it easier to determine the values that correspond to the displayed gradients (to do this, click **Colormap Options...** in the corresponding graphics object dialog box).

31.1.6. Displaying Results on a Sweep Surface

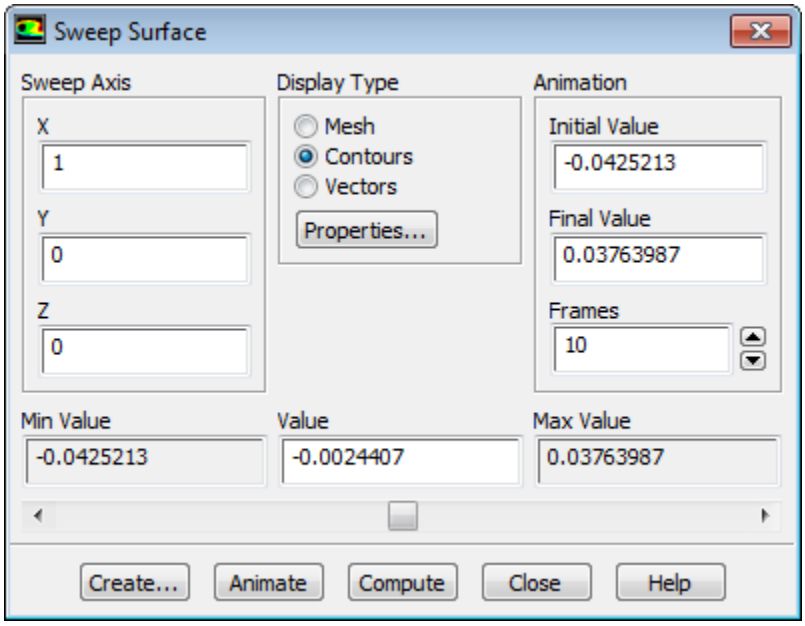
Sweep surfaces can be used when you want to examine the mesh, contours, or vectors on various sections of the domain without explicitly creating the corresponding surfaces. For example, if you want to display solution results for a 3D combustion chamber, instead of creating numerous surfaces at different cross-sections of the domain, you can use a sweep surface to view the variation of the flow and temperature throughout the chamber.

31.1.6.1. Steps for Generating a Plot Using a Sweep Surface

You can plot meshes, contours, or vectors on a sweep surface using the Sweep Surface Dialog Box (p. 2565) (Figure 31.27: The Sweep Surface Dialog Box (p. 1782)).

Results → Animations → Sweep Surface Edit...

Figure 31.27: The Sweep Surface Dialog Box



The basic steps for generating a mesh, contour, or vector plot using a sweep surface are as follows:

1. Under **Sweep Axis**, specify the (**X**, **Y**, **Z**) vector representing the axis along which the surface should be swept.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

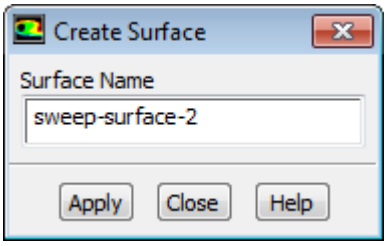
2. Click **Compute** to update the **Min Value** and **Max Value** to reflect the extents of the domain along the specified axis.

3. Under **Display Type**, specify the type of display you want to see: **Mesh**, **Contours**, or **Vectors**. The first time that you select **Contours** or **Vectors**, ANSYS Fluent will open the Contours Dialog Box (p. 2676) or the Vectors Dialog Box (p. 2831) so you can modify the settings for the display. To make subsequent modifications to the display settings, click **Properties** to open the Contours Dialog Box (p. 2676) or Vectors Dialog Box (p. 2831).

4. Move the slider under **Value** (which indicates the value of *x*, *y*, or *z*) to move the sweep surface through the domain along the specified **Sweep Axis**. ANSYS Fluent will update the mesh, contour, or vector display when you release the slider. You can also enter a position in the **Value** field and press **Enter** to update the display.

5. If you want to save the currently displayed sweep surface so that you can use it for a different type of plot (for example, a pathlines plot or an XY plot) or combine it with displays on other surfaces, click **Create...** to open the Create Surface Dialog Box (p. 2566) (Figure 31.28: The Create Surface Dialog Box (p. 1783)). Enter the **Surface Name** and click **OK**.

   Figure 31.28: The Create Surface Dialog Box

   The surface that is created is an isosurface based on the mesh coordinates; the contour or vector settings are not stored in the surface.

You can also animate the sweep surface display, as described below, rather than moving the slide bar yourself.

31.1.6.2. Animating a Sweep Surface Display

The steps for animating a sweep surface display are as follows:

1. Specify the **Sweep Axis** and **Display Type** as described above.

2. Under **Animation**, enter the **Initial Value** and **Final Value** for the animation. These values correspond to the minimum and maximum values along the **Sweep Axis** for which you want to animate the display.

3. Specify the number of **Frames** you want to see in the animation.

4. Click **Animate**.

31.1.7. Hiding the Graphics Window Display

There may be situations where displaying graphics on a local machine is not practical. Therefore, you may decide to hide (or disable) the graphics display window.

To disable the graphics display window when starting ANSYS Fluent from the command line, you can specify the driver as null:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
fluent -driver null
```

For an ANSYS Fluent session that is already in progress, the graphics window display can be disabled using the following TUI command:

`display → set → rendering-options → driver → null`

Important

All graphics windows must be closed prior to invoking the above TUI command.

If the graphics window display is disabled, you can continue to save graphics using the **Save Picture** option, as described in Saving Picture Files (p. 102). The saved graphics files will be identical whether the graphics window display is enabled or disabled.

For an ANSYS Fluent session that is already in progress, to re-enable a graphics window display that had been previously disabled, use the following TUI command:

`display → set → rendering-options → driver → opengl`

If any graphics windows are open (which are not visible to you), ANSYS Fluent will prompt you to close all open windows. You can close them using the following Scheme command:

```
(close-all-open-windows)
```

and then retype the TUI command to enable the graphics windows.

Important

If you happen to be logged on to a machine remotely, then `opengl` may not work on your system. Use `x11` instead to enable your graphics windows.

31.2. Customizing the Graphics Display

There are a number of ways in which you can alter the graphical display once you have generated the basic elements in it (contours, meshes, and so on). For example, you can overlay multiple graphics, add descriptive text or lighting to the plot, and modify the captions or legend layout. These and other customizations are described in this section.

31.2.1. Advanced Graphics Overlays
31.2.2. Opening Multiple Graphics Windows
31.2.3. Changing the Legend Display
31.2.4. Adding Text to the Graphics Window
31.2.5. Changing the Colormap
31.2.6. Adding Lights
31.2.7. Modifying the Rendering Options

Important

It is possible that the graphics window may become **[Out of Date]** (which would be indicated at the top of the graphics window), if you make changes to items that are already displayed. To resolve this out of date state, right-click in the graphics window and click **Refresh Display** in the context menu that appears.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*If a context menu does not appear on a right-click of the graphics window, ensure that the right-mouse button is set to an action other than **mouse-zoom** or **mouse-probe** and **long description** (**Viewing** ribbon tab, **Mouse** group box).*

31.2.1. Advanced Graphics Overlays

Normally, you can see only one picture at a time in the graphics window, unless you are Displaying a Scene (p. 1780). Sometimes, however, you may want to create an "exploded" view, where results translated or rotated out for the physical domain for enhanced display (see Figure 31.29: Exploded Scene Display of Temperature and Velocity (p. 1786)). You can do this by turning on the **Overlays** option (and clicking **Apply**) and using the transform operation in the Scene Description Dialog Box (p. 2577).

Viewing → Graphics → Compose...

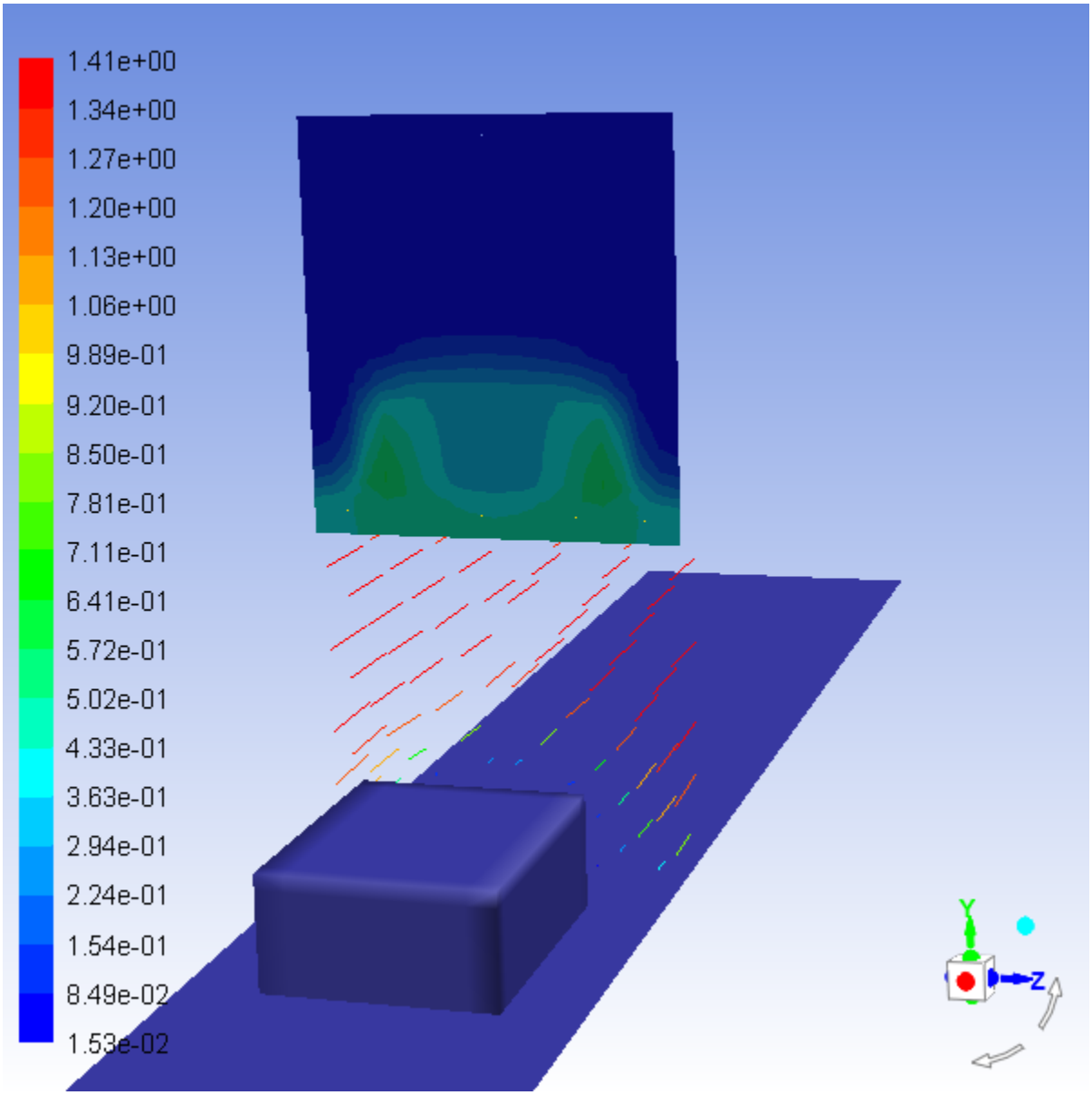
Once overlaying is enabled, subsequent graphics that you generate will be displayed on top of the existing display in the active graphics window. To generate a plot without overlays, you must turn off the **Overlays** option in the Scene Description Dialog Box (p. 2577) (and remember to click **Apply**).

When you are overlaying multiple graphics, the captions and color scale that will appear in the latest display are those that correspond to the most recently drawn graphic.

Note that when overlaying is enabled, it will apply to all graphics windows, including those that are not yet open. Turning overlays on and off does so for all graphics windows, not just for the active window. That is, if you enable overlays, open a new graphics window (as described in Opening Multiple Graphics Windows (p. 1786)), and then generate two or more graphics in that window, they will be overlaid.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.29: Exploded Scene Display of Temperature and Velocity



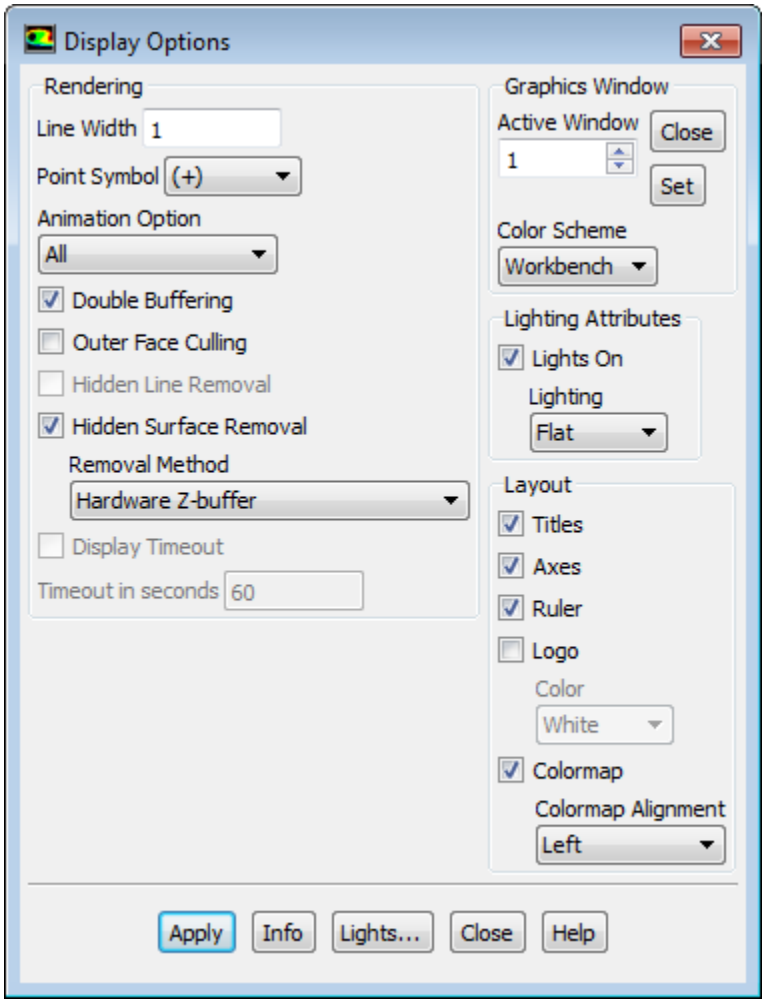
31.2.2. Opening Multiple Graphics Windows

During your ANSYS Fluent session, you can open up to 20 graphics windows at one time and they may be viewed within the application window or in separate windows. The windows are numbered 1 through 20 and the ID number for each window will appear at the top of the frame that surrounds it. You can view a specific window by selecting it from the drop-down list next to the ID number. The first time you display graphics, window 1 will be displayed automatically. To open an additional window, you can use the Display Options Dialog Box (p. 2573) (Figure 31.30: The Display Options Dialog Box (p. 1787)).

Results → Graphics Options...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.30: The Display Options Dialog Box



Use the up arrow to increment the window ID in the **Active Window** field under **Graphics Window** and then click **Open**. The **Close** button changes to the **Open** button if the **Active Window** is set to more than **1**.

To close an open window, increase or decrease the **Active Window** value to the ID of the window to be closed, and then click **Close** that appears next to the **Active Window** field. The **Open** button changes to a **Close** button if the **Active Window** is open.

To display a different color scheme for the graphics window, select **Classic** or **Workbench** from the drop-down list located below the **Active Window** field. All graphics windows will change.

31.2.2.1. Setting the Active Window

When you have more than one graphics window open, you must identify the active window so that ANSYS Fluent will know which one to draw the plot in. There are two ways to set the active window: you can simply click any mouse button in the desired graphics window, or you can specify the ID for the desired graphics window in the **Active Window** field (in the Display Options Dialog Box (p. 2573)) and click **Set**. Regardless of the method used, this window will remain active until you set a new active window.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.2.3. Changing the Legend Display

ANSYS Fluent graphics include, by default, a caption or legend block that consists of fields of text describing the contents of the graphic, the ANSYS Fluent product identification, an axis triad indicating the orientation of the displayed object, a color key defining the correspondence between each color and the magnitude of the plotted variable, and the ANSYS logo. You can turn off the display of the legend and color scale, and/or the axis triad. You can also hide the ANSYS logo. You can also display the colormap on any side of the display window as per convenience. In addition you can also edit the captions directly in the graphics window.

31.2.3.1. Enabling/Disabling the Legend, Logo, and Color Scale

You can disable the display of the **Titles**, **Axes**, **Logo**, and the **Colormap** scale by deselecting each of the options under **Layout** in the Display Options Dialog Box (p. 2573) (Figure 31.30: The Display Options Dialog Box (p. 1787)).

Results → Graphics Options...

Use the text interface to enable/disable the captions and color scale individually, and to change the size and position of the captions and color scale.

`display → set → windows → text →`

`display → set → windows → scale →`

See the separate Text Command List for details.

31.2.3.2. Editing the Legend

When captions are displayed in the graphics window, you may choose to modify, delete, or add to the text that appears in the caption box. To do so, click the left mouse button in the desired location. A cursor will appear, and you can then type new text or delete the text that was originally there (using the backspace or delete key). Note that changes to existing text in the caption block will be removed when you draw new graphics in the window (unless you are overlaying multiple graphics in the same window), but text that you add on a previously empty line in the caption block will not be removed until the default caption text makes use of that line.

31.2.3.3. Adding a Title to the Caption

You can define a title for your problem using the `title` text command:

`display → set → title`

The title you define will appear on the top line of the caption, at the far left, in all subsequent plots. It will also be saved in the case file.

Important

You will need to enclose your title in quotation marks (for example, " `my title` ").

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.2.3.4. Enabling/Disabling the Axes

You can disable the display of the axis triad by turning off the **Axes** option under **Layout** in the Display Options Dialog Box (p. 2573) (Figure 31.30: The Display Options Dialog Box (p. 1787)).

31.2.3.5. Enabling/Disabling the Ruler

You can disable the display of the ruler by turning off the **Ruler** option under **Layout** in the Display Options Dialog Box (p. 2573) (Figure 31.30: The Display Options Dialog Box (p. 1787)).

Note that if your project is 3D, then you must be in orthographic view in order to use the ruler.

31.2.3.6. Modifying and Displaying/Hiding the Logo

The graphics window displays a white ANSYS logo in the upper right corner. You may choose between a white (default) or black logo by selecting **White** or **Black** from the **Color** drop-down list under **Layout** in the Display Options Dialog Box (p. 2573) (Figure 31.30: The Display Options Dialog Box (p. 1787)).

You can prevent the ANSYS logo from being displayed in the graphics window by disabling the **Logo** option under **Layout**.

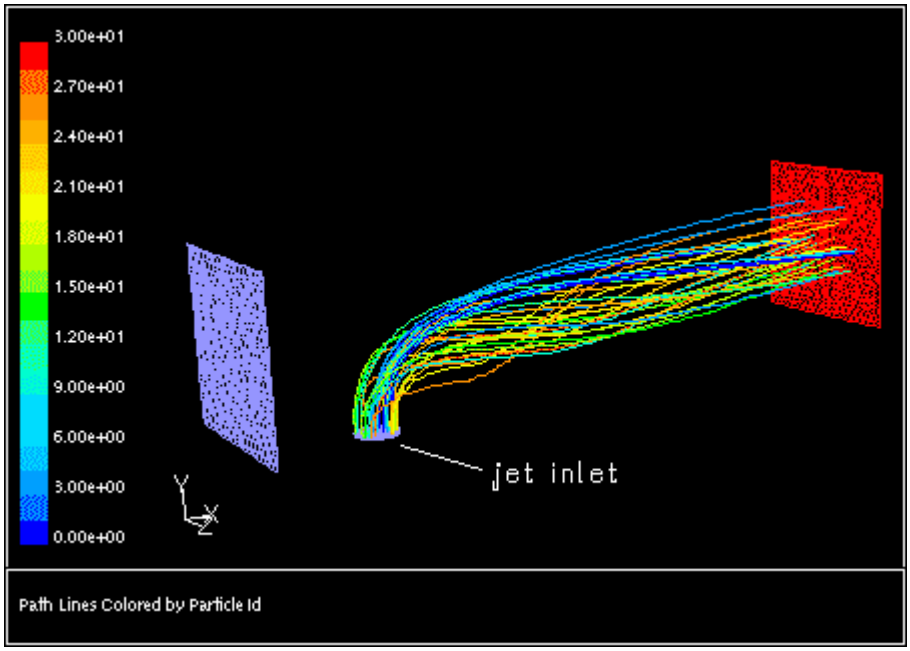
31.2.3.7. Colormap Alignment

You can set the position of the colormap on any side (left, top, bottom, or right) of the display window. Default alignment for the colormap is set to the **Left**. If you want to change the alignment, select the required direction in the **Colormap Alignment** drop-down list.

31.2.4. Adding Text to the Graphics Window

You can use the Annotate Dialog Box (p. 2592) to add text to the graphics window with optional attachment lines. Figure 31.31: Graphics Window with Text Annotation (p. 1789) shows an example of a graphics display with annotated text in it.

Figure 31.31: Graphics Window with Text Annotation

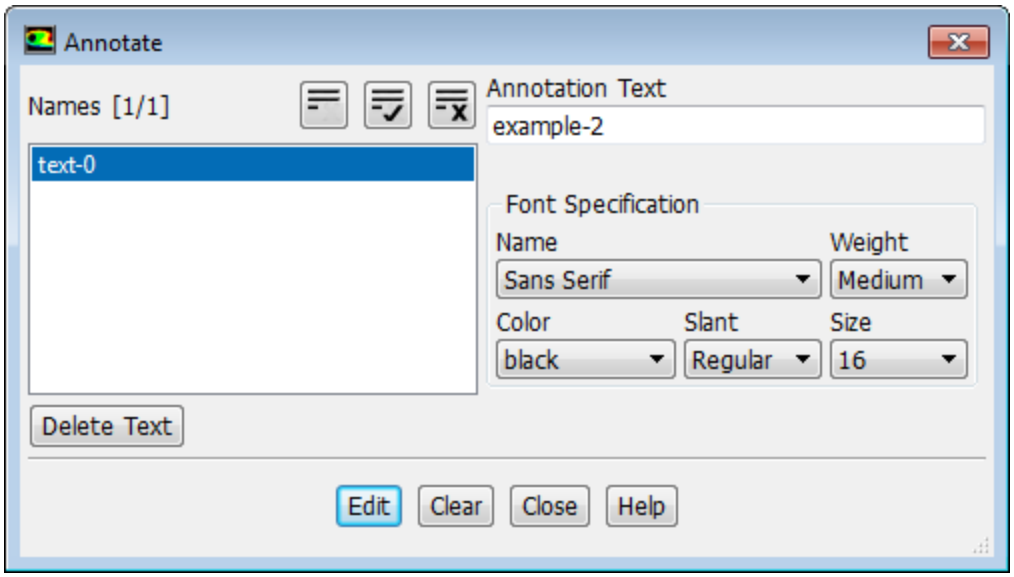


31.2.4.1. Adding Text Using the Annotate Dialog Box

Adding text to the graphics window using the Annotate Dialog Box (p. 2592) (Figure 31.32: The Annotate Dialog Box (p. 1790)) enables you to control the font and color of the text.

Results → Graphics Annotate...

Figure 31.32: The Annotate Dialog Box



The steps for adding text are as follows:

1. Under **Font Specification**, select the font type in the **Name** drop-down list, the font weight (**Medium** or **Bold**) in the **Weight** drop-down list, the size (in points) in the **Size** drop-down list, the color in the **Color** drop-down list, and the slant (**Regular** or **Italic**) in the **Slant** drop-down list.

2. Enter the text that you want to add in the **Annotation Text** field.

3. Click **Add**. You will be asked to pick the location in the graphics window where you want to place the text, using the mouse-probe button. By default, the mouse-probe button is the right button, but you can change this in the **Mouse** group box of the **Viewing** tab, as described in Controlling the Mouse Button Functions (p. 1800).

   If you click the mouse button once in the desired location, the text will be placed at that point. Dragging the mouse with the mouse-probe button depressed will draw an attachment line from the point where the mouse was first clicked to the point where it was released. The annotation text will be placed at the point where the mouse button was released.

31.2.4.2. Editing Existing Annotation Text

Once you have added text to the graphics display, you may change the font characteristics of one or more text items, or delete individual text items.

To modify or delete existing text, follow these steps:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the appropriate item in the **Names** list in the Annotate Dialog Box (p. 2592) (Figure 31.32: The Annotate Dialog Box (p. 1790)). When you select a name, the associated text will be displayed in the **Annotation Text** field, and the **Add** button becomes the **Edit** button.

2. Modify the **Font Specification** entries as desired, and click **Edit** to modify the text, or simply click **Delete Text** below the **Names** list to delete the selected text.

Note that if you want to make changes to all current annotation text, you can select all of the **Names** instead of just one in step 1.

You can move the text in the same way that you move other geometric objects in the display, using the Scene Description Dialog Box (p. 2577) and the Transformations Dialog Box (p. 2580). See Transforming Geometric Objects in a Scene (p. 1819) for details.

31.2.4.3. Clearing Annotation Text

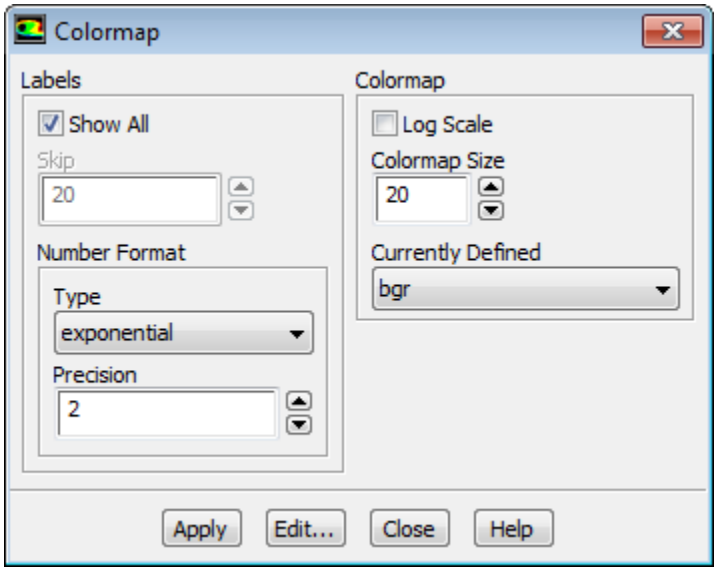
Annotation text is associated with the active graphics window and is removed only when the annotations are explicitly cleared. To remove the annotations from the graphics window, you must click **Clear** in the Annotate Dialog Box (p. 2592). If you draw new graphics in the window without clearing the annotations, they will remain visible in the new display.

31.2.5. Changing the Colormap

The default colormap used by ANSYS Fluent to display graphical data (for example, vectors) ranges from blue (minimum value) to red (maximum value). Additional predefined colormaps are available, and you can also create custom colormaps. To make any changes to the colormap, you will use the Colormap Dialog Box (p. 2589) (Figure 31.33: The Colormap Dialog Box (p. 1791)).

Results → Graphics Colormap...

Figure 31.33: The Colormap Dialog Box



When you plot contours, you can temporarily modify the number of colors in the colormap by changing the number of contour levels in the Contours Dialog Box (p. 2676); you will only need to use the Colormap Dialog Box (p. 2589) if you want to change other characteristics of the colormap.

Important

Note that if you are using a gray-scale colormap and you want to save a gray-scale picture, you should save a color picture. When you save a gray-scale picture, ANSYS Fluent uses an internal gray scale, not the gray scale specified by the colormap. If you save a color picture, the colormap you selected (that is, your gray scale) will be used.

31.2.5.1. Predefined Colormaps

The following colormaps are automatically available in ANSYS Fluent:

bgr:
Blue represents the minimum value, green the middle, and red the maximum value. Colors in between are interpolated from blue to green, and from green to red. (This is the default colormap.)

bgrb:
Blue represents the minimum and maximum values, and green and red are values 1/3 and 2/3 of the maximum value, respectively. Colors in between are interpolated from blue to green, from green to red, and from red to blue.

blue:
The minimum value is represented by blue-black, and the maximum value by pure blue.

cyan-yellow:
Cyan represents the minimum value and yellow represents the maximum value.

fea:
Blue represents the minimum value and red represents the maximum value. The colors in between are those used in third-party finite element analysis packages.

gray:
Black is used for the minimum value and white for the maximum value.

green:
The minimum value is represented by green-black, and the maximum value by pure green.

purple-magenta:
Purple represents the minimum value and magenta represents the maximum value.

red:
The minimum value is represented by red-black, and the maximum value by pure red.

rgb:
Red represents the minimum value, green the middle, and blue the maximum value. Colors in between are interpolated from red to green, and from green to blue.

The number of colors interpolated between the colors in the scale name (for example, between purple and magenta) will depend on the size of the colormap.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.2.5.2. Selecting a Colormap

The procedure for selecting a new colormap to be used in graphics displays is as follows:

1. In the Colormap Dialog Box (p. 2589) (Figure 31.33: The Colormap Dialog Box (p. 1791)), select the desired colormap in the **Currently Defined** drop-down list. This list will contain all of the colormaps predefined by ANSYS Fluent as well as any custom colormaps that you have created as described in Creating a Customized Colormap (p. 1795).

2. Set the colormap size and scale as described in Specifying the Colormap Size and Scale (p. 1793).

3. Click **Apply** to update the current graphics display with the new colormap. All future displays will use the newly selected colormap and options.

31.2.5.2.1. Specifying the Colormap Size and Scale

Once you have selected the desired colormap from the **Currently Defined** list, you may modify the **Colormap Size**. This value is the number of distinct colors in the color scale.

You can also choose to use a logarithmic scale instead of a decimal scale by turning on the **Log Scale** option. With a log scale, the color used in the graphics display will represent the log of the value at that location in the domain. The values represented by the colors will, therefore, increase exponentially.

31.2.5.2.2. Changing the Number Format

You can change the format of the labels that define the color divisions at the left of the graphics window using the controls under the **Number Format** heading in the Colormap Dialog Box (p. 2589).

- To display the real value with an integral and fractional part (for example, 1.0000), select **float** in the **Type** drop-down list. You can set the number of digits in the fractional part by changing the value of **Precision**.

- To display the real value with a mantissa and exponent (for example, 1.0e-02), select **exponential** in the **Type** drop-down list. You can define the number of digits in the fractional part of the mantissa in the **Precision** field.

- To display the real value with either float or exponential form, depending on the size of the number and the defined **Precision**, choose **general** in the **Type** drop-down list.

31.2.5.3. Displaying Colormap Label

You can customize the number of values displayed on the colormap. The default number of labels that appear alongside the colormap depends on the font size and the colormap size (Figure 31.34: The Default Colormap Label Display (p. 1794)).

- To reduce the number of labels that appear alongside the colormap, increase the number of labels skipped. To do so, deselect **Show All** in the Colormap Dialog Box (p. 2589) and set the number of labels to be skipped.

  To demonstrate what effect this command has on the display, enter a value of 4 under **Skip** (note that the value entered must be an integer). This will result in three intermediate labels being skipped, with the first and the last colormap values always being displayed (Figure 31.35: The Colormap with Skipped Labels (p. 1794)).

- To reset the original colormap display, simply select **Show All**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.34: The Default Colormap Label Display

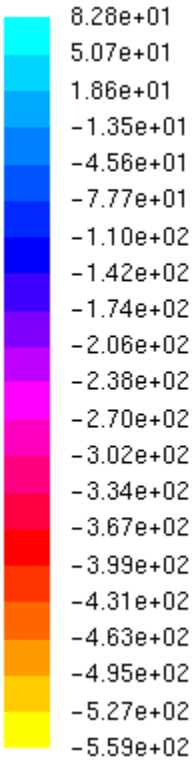
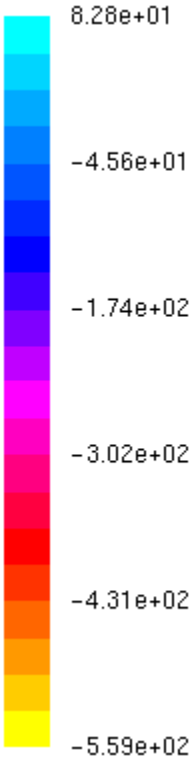


Figure 31.35: The Colormap with Skipped Labels



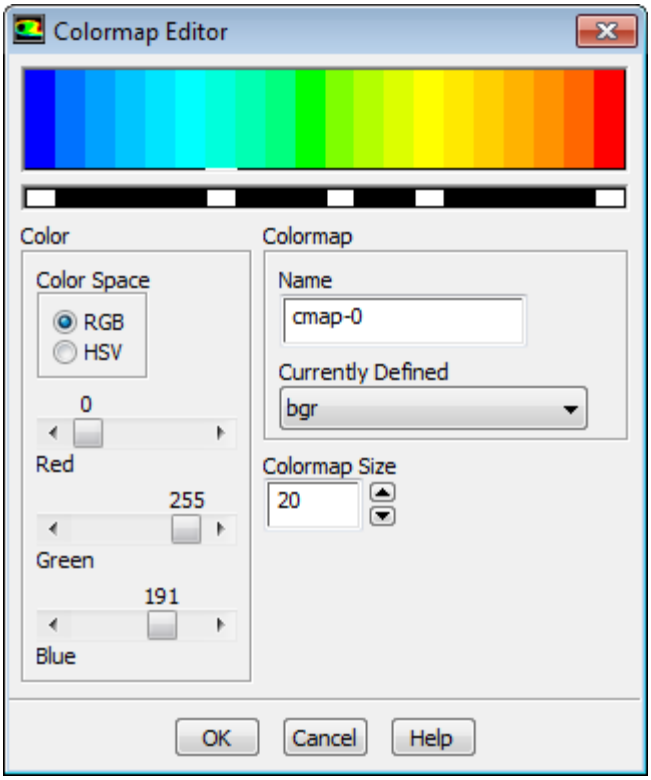
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

31.2.5.4. Creating a Customized Colormap

You can create your own colormap by manipulating the "anchor colors" and the colormap size. A color scale is created by linear interpolation between the anchor colors. The color, number, and position of the anchor colors will therefore control the description of the colormap. By increasing the colormap size, you can increase the total number of colors and obtain a color scale that changes more gradually. The procedure is as follows:

1. In the Colormap Dialog Box (p. 2589), click **Edit...** to open the Colormap Editor Dialog Box (p. 2591) (Figure 31.36: The Colormap Editor Dialog Box (p. 1795)).

   Figure 31.36: The Colormap Editor Dialog Box



2. In the Colormap Editor Dialog Box (p. 2591), select a color scale in the **Currently Defined** list as your starting point. The colors in the scale will be displayed at the top of the dialog box. A white bar below a color is an "anchor point" indicating that this color is an "anchor color".

3. If you want to add more colors to the color scale, increase the **Colormap Size**; to use fewer colors, decrease this value. When you use the counter arrows (or type in a value and press **Enter**), the color scale display at the top of the dialog box will be updated immediately.

   Important

   The total number of colors must not be less than the number of anchor points.

4. To obtain the desired color scale interpolation, manipulate the anchor colors as needed:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To add an anchor point, click any mouse button on the black space directly below the desired anchor color (or click the color itself). A white bar will appear below the color to identify it as an anchor color, and the color will automatically be selected for color-definition modification.
- To remove an anchor point, click the white bar below the anchor color. The white bar will disappear and the color scale will be updated to reflect the new interpolation.
- To select a current anchor color in order to modify its color definition, click the color itself at the top of the dialog box.
- To modify the color of the selected anchor color, you can change either the red/green/blue components (choose **RGB**, the default) or the hue/saturation/value components (choose **HSV**). **HSV** is recommended if you plan to record the graphics display on video, as it enables you to create a more subtle gradation of color and reduce the tendency of bright colors to "bleed". Move the **Red**, **Green**, and **Blue** or **Hue**, **Saturation**, and **Value** sliders to obtain the desired color. The color scale at the top of the dialog box will be updated automatically to show the effect of your change.

  Important

  It is a good idea to note the original value of a color component before moving the slider so that you will be able to return to it if you change your mind. (See Scales (p. 17) for instructions on using a scale slider.)

  If you make a mistake while modifying the color scale, you can start over by selecting the starting-point colormap in the **Currently Defined** list.

5. If you want to change the default name of the new colormap, enter the new name in the **Name** field. By default, custom colormaps are called `cmap-0`, `cmap-1`, and so on.
6. Click **OK** to save the new colormap. The colormap name appears in the **Currently Defined** list in the Colormap Dialog Box (p. 2589) and can be selected for use in the graphics display.

Custom colormap definitions will be saved in the case file.

31.2.6. Adding Lights

In ANSYS Fluent you can add lights with a specified color and direction to your display. These lights can enhance the appearance of the display when it contains 3D geometries. By default one light is defined. You can enable the effect of the existing light(s) using the Display Options Dialog Box (p. 2573) or the Lights Dialog Box (p. 2588), and you can add new lights using the Lights Dialog Box (p. 2588).

31.2.6.1. Turning on Lighting Effects with the Display Options Dialog Box

To enable the effect of lighting, you can use the Display Options Dialog Box (p. 2573).

Results → Graphics Options...

If you enable the **Lights On** option under **Lighting Attributes** and click **Apply**, you will see the lighting effects in the active graphics window. To turn off the lighting effects, simply turn off the **Lights On** option and click **Apply**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also choose the method to be used in lighting interpolation; select **Flat**, **Gouraud**, or **Phong** in the **Lighting** drop-down list. Flat is the most basic method: there is no interpolation within the individual polygonal facets. Gouraud and Phong have smoother gradations of color because they interpolate on each facet.

31.2.6.2. Turning on Lighting Effects with the Lights Dialog Box

You can also enable lighting effects using the Lights Dialog Box (p. 2588) (Figure 31.37: The Lights Dialog Box (p. 1797)).

Results → Graphics Lights...

For constant lighting effects in the direction of the view, enable the **Headlight On** option in the Lights Dialog Box (p. 2588). This option has the effect of a light source directly in front of the model, no matter what orientation the model is viewed in. To disable this feature, turn off the **Headlight On** option in the Lights Dialog Box (p. 2588).

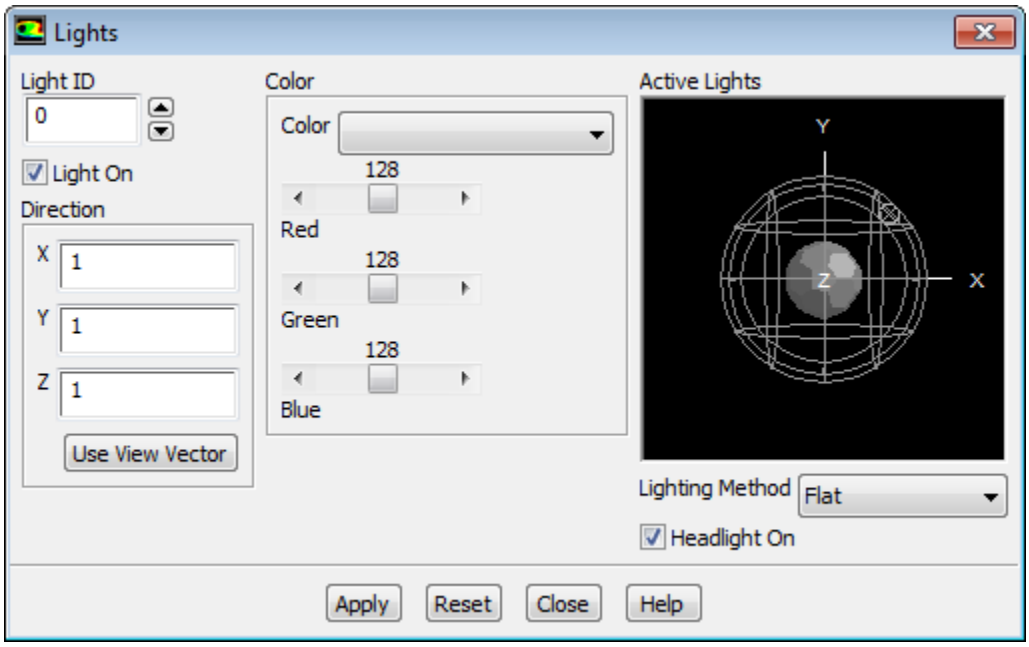
In the **Lighting Method** drop-down list, choose **Flat**, **Gouraud**, or **Phong** to enable the appropriate lighting method. These methods are described in the previous section. To disable lighting, select **Off** in the list. To see the lighting effects in the active graphics window, click **Apply**.

31.2.6.3. Defining Light Sources

You can control individual lights in the Lights Dialog Box (p. 2588) (Figure 31.37: The Lights Dialog Box (p. 1797)). The Lights Dialog Box (p. 2588) enables you to create a light and then turn it off without deleting it. In this way, you can retain lights that you have defined previously but do not want to use at present.

Results → Graphics Lights...

Figure 31.37: The Lights Dialog Box



(You can also open the Lights Dialog Box (p. 2588) by clicking **Lights...** in the Display Options Dialog Box (p. 2573).)

By default, light 0 is defined to be dark gray with a direction of (1,1,1). A light source is a distant light, similar to the sun. The direction (1,1,1) means that the rays from the light will be parallel to the vector from (1,1,1) to the origin. To create an additional light (for example, light 1), follow the steps listed below.

1. Increase **Light ID** to a new value (for example, 1).
2. Enable the **Light On** check box.
3. Define the light color by entering a descriptive string (for example, `lavender`) in the **Color** field, or by moving the **Red**, **Green**, and **Blue** sliders to obtain the desired color. The default color for all lights is dark gray.
4. Specify the light direction by doing one of the following:
   - Enter the (**X**, **Y**, **Z**) Cartesian components under **Direction**.
   - Click the middle mouse button in the desired location on the sphere under **Active Lights**. (You can also move the light along the circles on the surface of the sphere by dragging the mouse while holding down the middle button.) You can rotate the sphere by pressing the left mouse button and moving the mouse (like a trackball).
   - Use your mouse to change the view in the graphics window so that your position in reference to the geometry is the position from which you would like a light to shine. Then click **Use View Vector** to update the **X,Y,Z** fields with the appropriate values for your current position and update the graphics display with the new light direction. This method is convenient if you know where you want a light to be, but you are not sure of the exact direction vector.
5. Repeat steps 1–4 to add more lights.
6. When you have defined all the lights you want, click **Apply** to save their definitions.

31.2.6.3.1. Removing a Light

To remove a light, enter the ID number of the light to be removed in the **Light ID** field and then clear the **Light On** check box. When a light is turned off, its definition is retained, so you can easily add it to the display again at a later time by selecting the **Light On** check box. For example, you may want to define three different lights to be used in different scenes. You can define each of them, and then enable only one or two at a time, using the **Light ID** field and the **Light On** check box. Once you have made all the desired modifications to the lights, remember to click **Apply** to save the changes.

31.2.6.3.2. Resetting the Light Definitions

If you have made changes to the light definitions, but you have not yet clicked **Apply**, you can reset the lights by clicking **Reset**. All lighting characteristics will revert to the last saved state (that is, the lighting that was in effect the last time you opened the dialog box or clicked on **Apply**).

31.2.7. Modifying the Rendering Options

Depending on the objects in your display window and what kind of graphics hardware and software you are using, you may want to modify some of the rendering parameters listed below. All are listed

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

under the **Rendering** heading in the Display Options Dialog Box (p. 2573) (Figure 31.30: The Display Options Dialog Box (p. 1787)).

Results → Graphics Options...

After making a change to any of these rendering parameters, click **Apply** to re-render the scene in the active graphics window with the new attributes. To see the effect of the new attributes on another graphics window, you must redisplay it or make it the active window (see Setting the Active Window (p. 1787)) and click **Apply** again.

Line Width:

By default, all lines drawn in the display have a thickness of 1 pixel. If you want to increase the thickness of the lines, increase the value of **Line Width**.

Point Symbol:

By default, nodes displayed on surfaces and data points on line or rake surfaces are represented in the display by a + sign inside a circle. If you want to modify this representation (for example, to make the nodes easier to see), you can select a different symbol in the **Point Symbol** drop-down list.

Animation Options:

There are two animation options that you can choose from. They are as follows:

All

uses a solid-tone shading representation of all geometry during mouse manipulation.

Wireframe

uses a wireframe representation of all geometry during mouse manipulation. If your computer has a graphics accelerator, you may not want to use this option; otherwise, the mouse manipulation may be very slow.

Double Buffering:

Enabling the **Double Buffering** option can dramatically reduce screen flicker during graphics updates. Note, however, that if your display hardware does not support double buffering and you turn this option on, double buffering will be done in software. Software double buffering uses extra memory.

Outer Face Culling:

This option enables you to turn off the display of outer faces in wall zones. **Outer Face Culling** is useful for displaying both sides of a slit wall. By default, when you display a slit wall, one side will "bleed" through to the other. When you enable the **Outer Face Culling** option, the display of a slit wall will show each side distinctly as you rotate the display. This option can also be useful for displaying two-sided walls (that is, walls with fluid or solid cells on both sides).

Hidden Line Removal:

If you do not use hidden line removal, ANSYS Fluent will not try to determine which lines in the display are behind others; it will display all of them, and a cluttered display will result for most 3D mesh displays. For most 3D problems, therefore, you should enable the **Hidden Line Removal** option. You should turn this option off (for optimal performance) if you are working with a 2D problem or with geometries that do not overlap.

Hidden Surface Removal:

If you do not use hidden surface removal, ANSYS Fluent will not try to determine which surfaces in the display are behind others; it will display all of them, and a cluttered display will result for most 3D mesh displays. For most 3D problems, therefore, you should enable the **Hidden Surface Removal** option. You

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

should turn this option off (for optimal performance) if you are working with a 2D problem or with geometries that do not overlap.

You can choose one of the following methods for performing hidden surface removal in the **Hidden Surface Method** drop-down list. These options vary in speed and quality, depending on the device you are using.

Hardware Z-buffer
is the fastest method if your hardware supports it. The accuracy and speed of this method is hardware-dependent. Note that if this method is not available on your computer, selecting it will cause the **Software Z-buffer** method to be used.

Painters
will show fewer edge-aliasing effects than **Hardware-Z-buffer**. This method is often used instead of **Software-Z-buffer** when memory is limited.

Software Z-buffer
is the fastest of the accurate software methods available (especially for complex scenes), but it is memory-intensive.

Z-sort only
is a fast software method, but it is not as accurate as **Software-Z-buffer**.

31.2.7.1. Graphics Device Information

If you need to know which graphics driver you are using and what graphics hardware it recognizes, you can click **Info** in the Display Options Dialog Box (p. 2573). The graphics device information will be printed in the text (console) window.

31.3. Controlling the Mouse Button Functions

A convenient feature of ANSYS Fluent is that it enables you to assign a specific function to each of the mouse buttons. According to your specifications, clicking a mouse button in the graphics window will cause the appropriate action to be taken. These functions apply only to the graphics windows; they behave differently when an XY plot or histogram is displayed. For information about the use of mouse buttons in these plots, see Plot Types (p. 1838). Clicking any mouse button in a graphics window will make that window the active window.

Important

3Dconnexion Space products (Mouse, Pilot, and Navigator) are not supported with ANSYS Fluent.

For additional information, see the following sections:
31.3.1. Button Functions
31.3.2. Modifying the Mouse Button Functions

31.3.1. Button Functions

The predefined button functions available are listed below:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

mouse-rotate
Enables you to rotate the view by dragging the mouse across the screen. Dragging horizontally rotates the object about the screen's y-axis; vertical mouse movement rotates the object about the screen's x-axis. The function completes when the mouse button is released or the cursor leaves the graphics window.

mouse-dolly
Enables you to translate the view by dragging the mouse while holding down the button. The function completes when the mouse button is released or the cursor leaves the graphics window.

mouse-scroll-zoom (non-modifiable)
Enables you to use the middle scroll-wheel to zoom in and out on the graphics window based on the location of the mouse pointer.

mouse-zoom
Enables you to draw a zoom box, anchored at the point at which the button was pressed, by dragging the mouse with the button held down. When you release the button, if the dragging was from left to right, a magnified view of the area within the zoom box will fill the window. If the dragging was from right to left, the area of the window is shrunk to fit into the zoom box, resulting in a "zoomed out" view. If the mouse button is simply clicked (not dragged), the selected point becomes the center of the window.

mouse-roll-zoom
Enables you to rotate or zoom the view, depending on the direction in which you drag the mouse. If you drag the mouse horizontally, the display will rotate about the axis normal to the screen. If you drag it vertically, the display will be magnified (if you drag it down) or shrunk (if you drag it up). The function completes when the mouse button is released or the cursor leaves the graphics window.

mouse-probe
Enables you to select items from the graphics windows and request information about displayed scenes. If you have selected **short description** from the **Probe** drop-down list and then click the mouse-probe button in the graphics window, only the identity of the item on which you clicked will be printed out in the console; if you have selected **long description** instead, more detailed information about a selected item will be displayed.

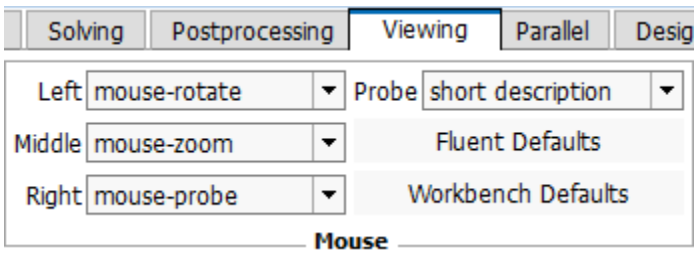
Note

A left-click will always select unless the left-mouse-button is set as **mouse-zoom** or **mouse-probe**.

31.3.2. Modifying the Mouse Button Functions

Mouse button functions are specified in the **Viewing** tab.

Viewing



For each mouse button (**Left**, **Middle**, and **Right**), select the desired function in the drop-down list. The functions are listed above. If you assign the probe function to one of the buttons, select **short description** or **long description** from the **Probe** drop-down list to indicate the amount of detail you want displayed in the console.

The new button functions take effect as soon as they are changed. That is, you do not have to redraw the graphics window to use the new functions; the appropriate function will be executed when a mouse button is subsequently clicked in a graphics window.

The **ANSYS Fluent Defaults** button functions are as follows:

Mouse Button	Key	Operation	2D Action	3D Action
Left		Drag	**Pan/dolly**	**Rotate**
Left		Click	**Select**	**Select**
Left	**Ctrl**	Click	**Add or remove selection**	**Add or remove selection**
Middle		Scroll	**Zoom**	**Zoom**
Middle		Drag	**Box-zoom**	**Box-zoom**
Middle	**Shift**	Drag	**Box-zoom**	**Box-zoom**
Middle	**Ctrl**	Drag	**Pan/dolly**	**Pan/dolly**
Right		Click	**Probe/context menu**	**Probe/context menu**
Right		Drag	**Probe/context menu**	**Probe/context menu**
Right	**Shift**	Click	**Probe/context menu**	**Probe/context menu**

The **Workbench Defaults** button functions are as follows:

Mouse Button	Key	Operation	2D Action	3D Action
Left		Drag	**Pan/dolly**	**Pan/dolly**
Left		Click	**Select**	**Select**
Left	**Ctrl**	Click	**Add or remove selection**	**Add or remove selection**
Middle		Scroll	**Zoom**	**Zoom**
Middle		Drag	**Probe**	**Rotate**
Middle	**Shift**	Drag	**Probe**	**Box-zoom**
Middle	**Ctrl**	Drag	**Probe**	**Pan/dolly**
Right		Click	**Context menu**	**Context menu**
Right		Drag	**Zoom**	**Zoom**
Right	**Shift**	Click	**Fit to center**	**Fit to center**

31.4. Viewing the Application Window

In ANSYS Fluent, the application window houses the ribbon and console, as well as multiple graphics windows, task pages, and the tree. By default, all components are displayed and one graphics window is visible. You can toggle the visibility of the toolbars, task page, console, and graphics window using in the standard toolbar. The ribbon and the tree are never fully hidden—you can hide the ribbon

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

contents using (also in the standard toolbar), but the tab names are always visible. You also have the option of viewing separate graphics windows as described in Opening Multiple Graphics Windows (p. 1786). For additional information about the various elements of the application window, see Graphical User Interface (GUI) (p. 1).

31.5. Modifying the View

ANSYS Fluent enables you to select and control the view of the scene that is displayed in the graphics window. You can modify the view by scaling, centering, rotating, translating, or zooming the display. You can also save a view that you have created, or restore or delete a view that you saved earlier. These operations are performed in the Views Dialog Box (p. 2583) (Figure 31.39: The Views Dialog Box (p. 1805)) or with the mouse, or simply using the shortcut in the graphics toolbar, as described in Selecting a View (p. 1803).

Important

You can revert to the previous view by pressing **Ctrl**+**L** while the graphics window has the main focus. You can also use the text command `view last` from the top level of the text command tree. You can use the command to revert to any of the past 20 views.

For additional information, see the following sections:

31.5.1. Selecting a View
31.5.2. Manipulating the Display
31.5.3. Controlling Perspective and Camera Parameters
31.5.4. Saving and Restoring Views
31.5.5. Mirroring and Periodic Repeats

31.5.1. Selecting a View

You can use the Views Dialog Box (p. 2583) (Figure 31.39: The Views Dialog Box (p. 1805)) to select the orientation of your display or use the drop-down available in the graphics toolbar for 3D simulations. Furthermore, you can simply click the interactive triad (Figure 31.38: Using the Triad to Change the Orientation of the Object (p. 1804)), displayed in the graphics window. The Table 31.1: Standard Views (p. 1803) relates the **Views** listed in the Views Dialog Box (p. 2583) to the views available in the graphics toolbar (see Pointer Tools (p. 7)).

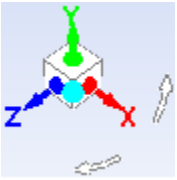
Results → Graphics → Views...

Table 31.1: Standard Views

View Listed in Dialog Box	View Shortcut in Graphics Toolbar
back	
bottom	
front	

View Listed in Dialog Box	View Shortcut in Graphics Toolbar
isometric	
left	
right	
top	

Figure 31.38: Using the Triad to Change the Orientation of the Object



To change the orientation of the object in the graphics window using the triad (Figure 31.38: Using the Triad to Change the Orientation of the Object (p. 1804)), you can

- Click an axis/semi-sphere to point it in the positive/negative direction.
- Right-click an axis/semi-sphere to point it in the negative/positive direction.
- Click the cyan iso-ball to set the isometric view.
- Click the white rotational arrows at the bottom of the triad to perform in-plane clockwise or counterclockwise 90 degree rotations.
- Left-click and hold—in the vicinity of the triad—and use the mouse to perform free-rotations in any direction. Release left mouse button to stop rotating.

Note that clicking on the X-axis will result in the +X-axis pointing towards you.

Note

For 2D cases, the drop-down and the interactive triad are not available; in a front view, the X-axis is horizontal with the positive direction to the right, the Y-axis is vertical with the positive direction pointing up, and the Z-axis is determined by the right-hand rule.

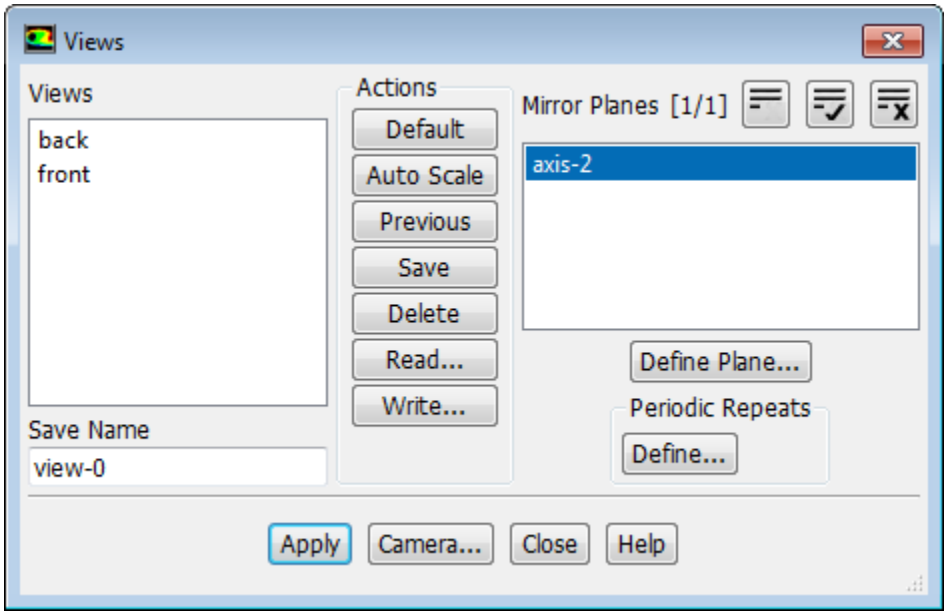
31.5.2. Manipulating the Display

Most of the manipulating activities (like scaling and centering the display) are accomplished using the Views Dialog Box (p. 2583) (Figure 31.39: The Views Dialog Box (p. 1805)).

Results → Graphics Views...

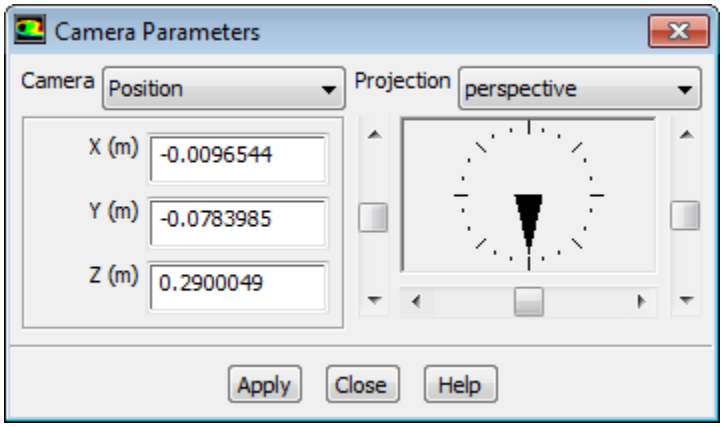
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.39: The Views Dialog Box



You can rotate, translate, and zoom the graphics display using either the mouse or the Camera Parameters Dialog Box (p. 2587) (Figure 31.40: The Camera Parameters Dialog Box (p. 1805)), which is opened from the Views Dialog Box (p. 2583).

Figure 31.40: The Camera Parameters Dialog Box



31.5.2.1. Scaling and Centering

You can scale and center the current display without changing its orientation by clicking **Auto Scale** in the Views Dialog Box (p. 2583).

31.5.2.2. Rotating the Display

Use either the mouse or the Camera Parameters Dialog Box (p. 2587) to:

- Rotate the display in any direction (in 3D)
- Rotate the display about the axis normal to the screen (in 2D)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To rotate a display with the mouse, use the button with the **mouse-rotate** function (the left button, by default). For information about changing the mouse functions, see Controlling the Mouse Button Functions (p. 1800).

- Click and drag the left mouse button in the graphics window to rotate the geometry in the display.
- You can also click and drag the left mouse button on the (x, y, z) graphics triad in the lower left corner to rotate the display.
- If you press the **Shift** key when you first click the mouse button to begin the rotation, the rotation will be constrained to a single direction (for example, you can rotate about the screen's horizontal axis without changing the position relative to the vertical axis).
- If you want to constrain the rotation of a display to be about the axis normal to the screen, you can also use the **mouse-roll-zoom** function.
- Click the appropriate mouse button and drag the mouse to the left for clockwise rotation, or to the right for counterclockwise rotation.

To rotate a 3D display using the Camera Parameters Dialog Box (p. 2587) (Figure 31.40: The Camera Parameters Dialog Box (p. 1805)) use the dial and the slider of the scales.

- To rotate about the horizontal axis at the center of the screen, move the slider on the scale to the left of the dial up or down (see Scales (p. 17) or instructions on using the scale).
- To rotate about the vertical axis at the center of the screen, move the slider on the scale below the dial to the left or right.
- To rotate about the axis at the center of and perpendicular to the screen, click the left mouse button on the indicator in the dial and drag it around the dial.

Important

The position of the slider or the dial indicator does not reflect the cumulative rotation about the axis as the slider/indicator will return to its original position when you release the mouse button.

31.5.2.2.1. Spinning the Display with the Mouse

When you use the mouse for rotation, you have the option to "push" the display into a continuous spin. This feature can be used in conjunction with video recording, or simply for interactive viewing of the domain from different angles. To use this option, use the `auto-spin?` text command:

`display` → `set` → `rendering-options` → `auto-spin?`

Then display the graphics (or, if the graphics are already displayed, you can click **Apply** in the Display Options Dialog Box (p. 2573)). The **mouse-rotate** button will then have two uses:

- To perform the standard rotation, stop dragging the mouse before you release the **mouse-rotate** button.
- To start the continuous spin, release the **mouse-rotate** button while you are still dragging the mouse. The display will continue to rotate on its own until you click any mouse button in the graphics window again. The speed of the rotation will depend on how fast you are dragging the mouse when you release the button.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For smoother rotation, enable the **Double Buffering** option in the Display Options Dialog Box (p. 2573) (see Modifying the Rendering Options (p. 1798)). This will reduce screen flicker during graphics updates.

31.5.2.3. Translating the Display

By default the left mouse button is set to **mouse-dolly** in 2D. For information about changing the mouse functions, see Controlling the Mouse Button Functions (p. 1800). Click and drag the left mouse button in the graphics window to translate the geometry in the display.

In 3D, you can either change one of the button functions to **mouse-dolly** and follow the instructions above for 2D, or use the **mouse-zoom** button (the middle button by default). Click the middle button once on the point in the display that you want to move to the center of the screen. ANSYS Fluent will redisplay the graphic with that point in the center of the window. This method can also be used in 2D.

31.5.2.4. Zooming the Display

In both 2D and 3D you will use the mouse button with the **mouse-zoom** function (the middle button by default) or the **mouse-roll-zoom** function (see Controlling the Mouse Button Functions (p. 1800) for information about enabling this optional function), or the Camera Parameters Dialog Box (p. 2587) to magnify and shrink the display.

With the **mouse-zoom** function, click the middle mouse button and drag it from left to right (creating a "zoom box") to magnify the display. Figure 31.41: Zooming In (Magnifying the Display) (p. 1807) displays the correct dragging of the mouse, from upper left to lower right on the display, in order to zoom. You can also drag from lower left to upper right. After you release the mouse button, ANSYS Fluent will re-display the graphic, filling the graphics window with the portion of the display that previously occupied the zoom box.

Figure 31.41: Zooming In (Magnifying the Display)

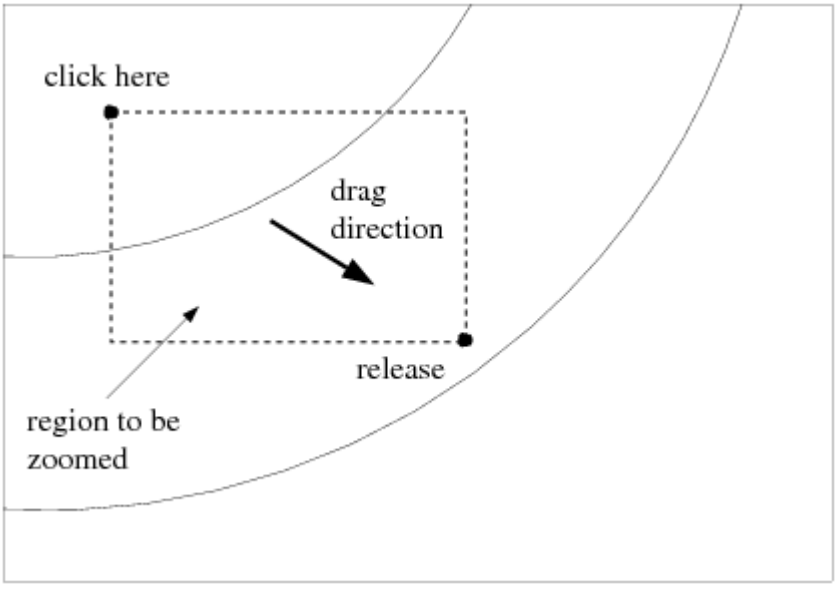
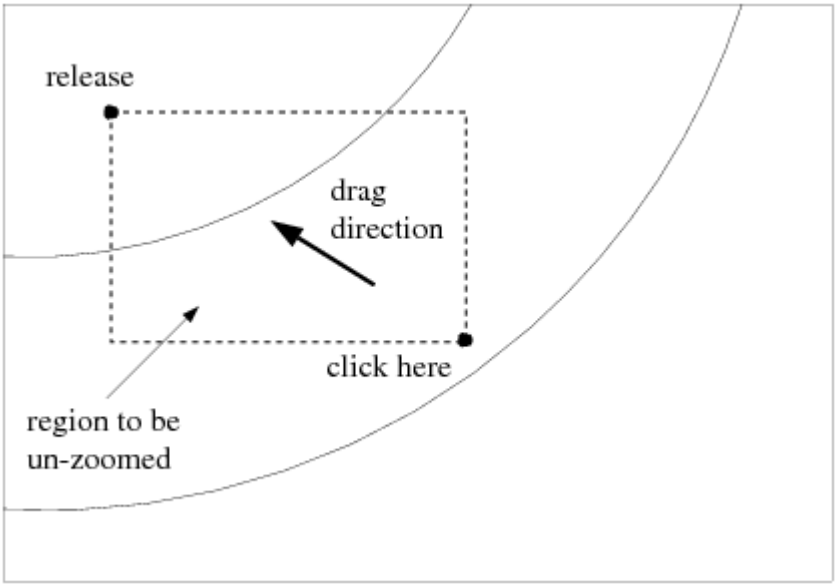


Figure 31.42: Zooming Out (Shrinking the Display)



Click the middle mouse button and drag it from right to left to shrink the display. Figure 31.42: Zooming Out (Shrinking the Display) (p. 1808) displays the correct dragging of the mouse, from lower right to upper left on the display, in order to "zoom out". You can also drag from upper right to lower left. After you release the mouse button, ANSYS Fluent will redisplay the graphic, shrinking the graphical display by the ratio of sizes of the zoom box you created and the previous display.

With the **mouse-roll-zoom** function, click the appropriate mouse button and drag the mouse down to zoom in continuously, or up to zoom out. In the Camera Parameters Dialog Box (p. 2587) (Figure 31.40: The Camera Parameters Dialog Box (p. 1805)), use the scale to the right of the dial to zoom the display. Move the slider bar up to zoom in and down to zoom out.

31.5.3. Controlling Perspective and Camera Parameters

Perspective and other camera parameters are defined in the Views Dialog Box (p. 2583), which you can open by clicking **Camera...** in the Camera Parameters Dialog Box (p. 2587) (Figure 31.39: The Views Dialog Box (p. 1805)).

Results → Graphics Views...

31.5.3.1. Perspective and Orthographic Views

You may choose to display either an orthographic view or a perspective view of your graphics. To show a perspective view (the default), select **Perspective** in the **Projection** drop-down list in the Views Dialog Box (p. 2583) (Figure 31.40: The Camera Parameters Dialog Box (p. 1805)). To turn off perspective, select **Orthographic** in the **Projection** drop-down list.

31.5.3.2. Modifying Camera Parameters

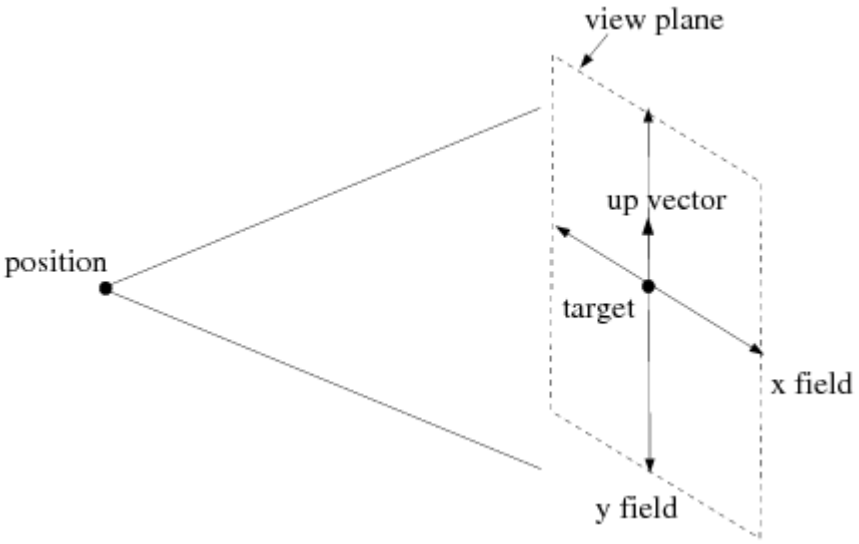
Instead of translating, rotating, and zooming the display as described in Manipulating the Display (p. 1804), you may sometimes want to modify the “camera” through which you are viewing the graphics display.

The camera is defined by four parameters: position, target, up vector, and field, as illustrated in Figure 31.43: Camera Definition (p. 1809). "Position" is the camera’s location. "Target" is the location of the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

point the camera is looking at, and "up vector" indicates to the camera which way is up. "Field" indicates the field of view (width and height) of the display.

Figure 31.43: Camera Definition



To modify the camera's position, select **Position** in the **Camera** drop-down list and specify the **X**, **Y**, and **Z** coordinates of the desired point. To modify the target location, select **Target** in the **Camera** drop-down list and specify the coordinates of the desired point. Select **Up Vector** to change the up direction. The up vector is the vector from (0,0,0) to the specified (**X,Y,Z**) point. Finally, to change the field of view, select **Field** in the **Camera** list and enter the **X** (horizontal) and **Y** (vertical) field distances.

Important

Click **Apply** after you change each camera parameter (**Position**, **Target**, **Up Vector**, and **Field**).

31.5.4. Saving and Restoring Views

After you make changes to the view shown in your graphics window, you may want to save the view so that you can return to it later. Several default views are predefined for you, and can be easily restored. All saving and restoring functions are performed with the Views Dialog Box (p. 2583) (Figure 31.39: The Views Dialog Box (p. 1805)).

Results → Graphics Views...

Important

Note that settings for mirroring and periodic repeats are not saved in a view.

31.5.4.1. Restoring the Default View

When experimenting with different view manipulation techniques, you may accidentally "lose" your geometry in the display. You can easily return to the default (front) view by clicking **Default** in the Views Dialog Box (p. 2583).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.5.4.2. Returning to Previous Views

After manipulating the display and viewing it from different angles, you can return to previous displays by clicking **Previous** in the Views Dialog Box (p. 2583).

31.5.4.3. Saving Views

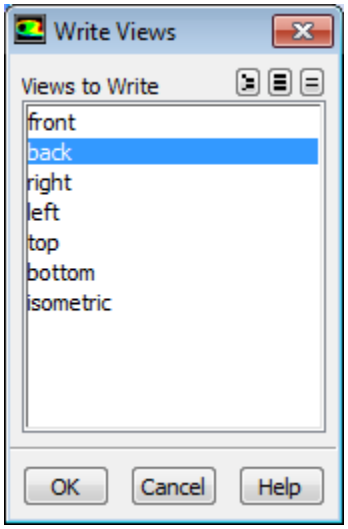
Once you have created a new view that you want to save for future use, enter a name for it in the **Save Name** field in the Views Dialog Box (p. 2583) and click **Save**. Your new view will be added to the list of **Views**, and you can restore it later as described below.

If a view with the same name already exists, you will be asked in a Question Dialog Box (p. 18) if it is OK to overwrite the existing view. If you overwrite one of the default views (top, left, right, front, and so on), be sure to save it in a view file if you want to use it in a later session. Although all views are saved to the case file, the default views are recomputed automatically when a case file is read in. Any custom view with the same name as a default view will be overwritten at that time.

As mentioned previously, all defined views will be saved in the case file when you write one. If you plan to use your views with another case file, you can write a "view file" containing just the views. You can read this view file into another Fluent session involving a different case file and restore any of the defined views, as described below.

To save a view file, click **Write...** in the Views Dialog Box (p. 2583). In the resulting Write Views Dialog Box (p. 2584) (Figure 31.44: The Write Views Dialog Box (p. 1810)), select the views you want to save in the **Views to Write** list and click **OK**. You will then use the **Select File** dialog box to specify the file name and save the view file. See The Select File Dialog Box (p. 19) for more details.

Figure 31.44: The Write Views Dialog Box



31.5.4.4. Reading View Files

If you have saved views to a view file (as described above), you can read them into your current Fluent session by clicking **Read...** in the Views Dialog Box (p. 2583), and indicating the name of the view file in The Select File Dialog Box (p. 19). If a view that you read has the same name as a view that already exists, you will be asked in a Question Dialog Box (p. 18) if it is OK to overwrite (that is, replace) the existing view.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

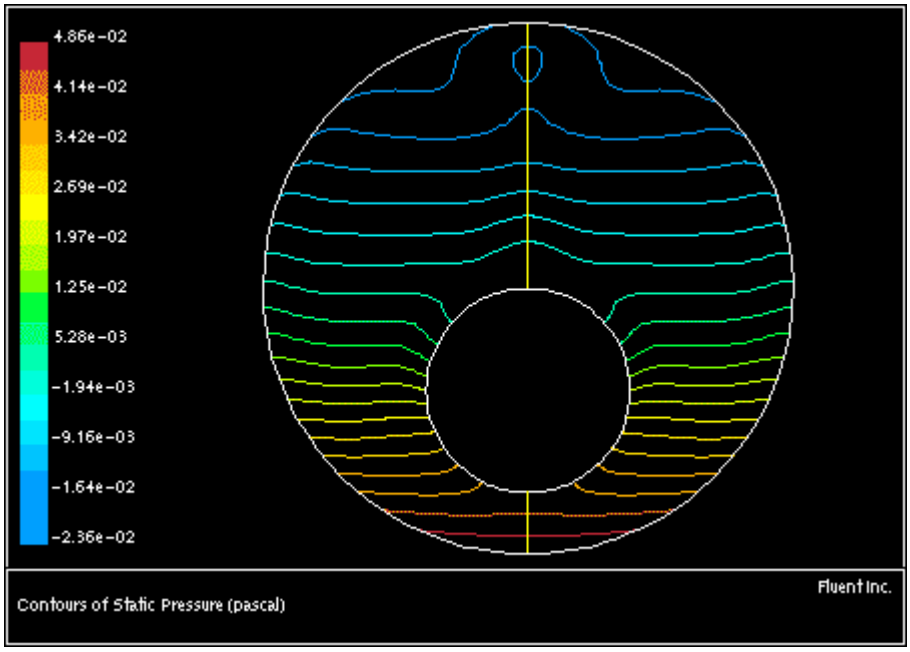
31.5.4.5. Deleting Views

If you decide that you no longer want to keep a particular view, you can delete it by selecting it in the **Views** list and clicking on **Delete**. Use this option carefully, so that you do not accidentally delete one of the predefined views.

31.5.5. Mirroring and Periodic Repeats

If you model the problem domain as a subset of the complete geometry using symmetry or periodic boundaries, you can display results on the complete geometry by mirroring or repeating the domain. For example, only one half of the annulus shown in Figure 31.45: Mirroring Across a Symmetry Boundary (p. 1811) was modeled, but the graphics are displayed on both halves. You can also define mirror planes or periodic repeats just for graphical display, even if you did not model your problem using symmetry or periodic boundaries.

Figure 31.45: Mirroring Across a Symmetry Boundary

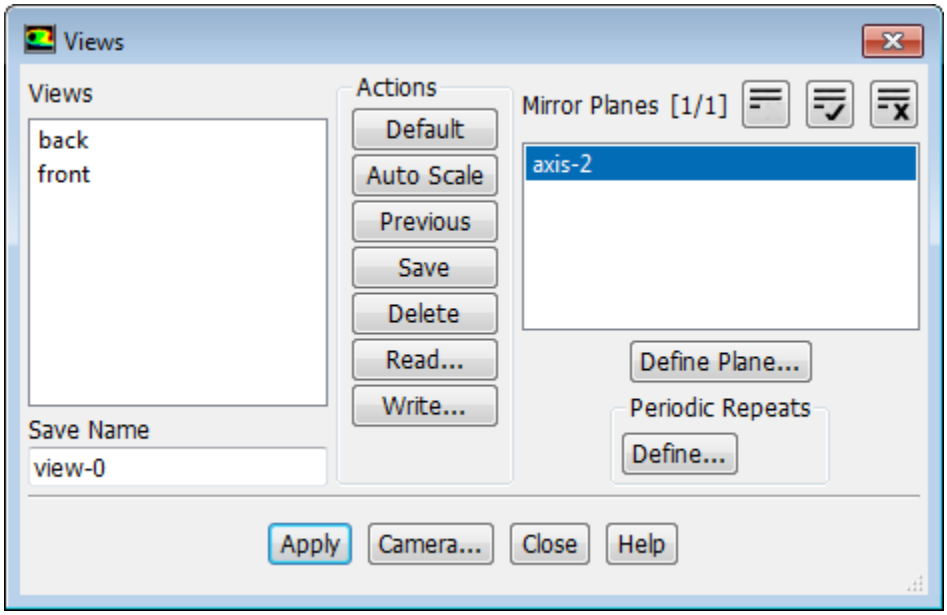


Display of symmetry and periodic repeats is controlled in the Views Dialog Box (p. 2583) (Figure 31.46: The Views Dialog Box (p. 1812)).

Results → Graphics Views...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.46: The Views Dialog Box



For a symmetric domain, all symmetry boundaries are listed in the **Mirror Planes** list. Select one or more of these boundaries as the plane(s) about which to mirror the display.

For a periodic domain, click **Define...** to open the Graphics Periodicity Dialog Box (p. 2586), to access the periodicity parameters. Specify the number of times to repeat the modeled portion by increasing the value of **Number of Repeats**. If, for example, you modeled a 90° sector of a duct and you wanted to display results on the entire duct, you would set **Number of Repeats** to 4.

In some cases, there may be multiple zones with different periodicity in the domain. For example, in turbomachinery problems with multiple blade rows using the mixing plane model, the periodic angles are different for each blade row. One blade may contain 20 blades (18° periodic angle) and other may contain 15 blades (24° periodic angle). In such cases select the required cell zone and specify the number of repeats for that particular cell zone.

When you click **Set** in the Graphics Periodicity Dialog Box (p. 2586) the graphics display will be immediately updated to show the requested periodic repeats.

Figure 31.47: Before Applying Periodicity (p. 1813) and Figure 31.48: After Applying Periodicity (p. 1813) shows the display for the sample geometry before and after applying the periodic repeats respectively. In this case the value of **Number of Repeats** is set to 6 for the 60° sector (outer part) and to a value of 4 is set for the 90° sector (inner part) of the geometry.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.47: Before Applying Periodicity

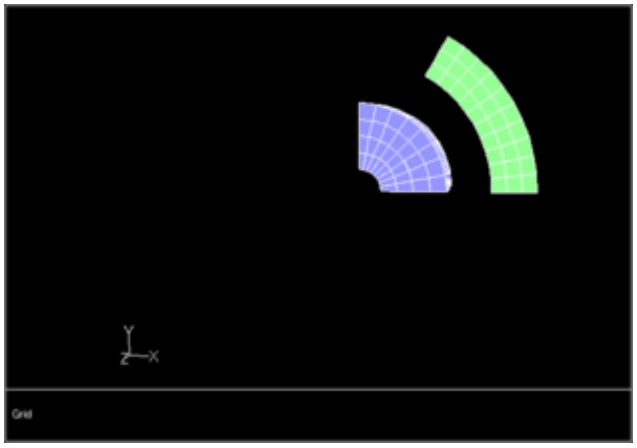
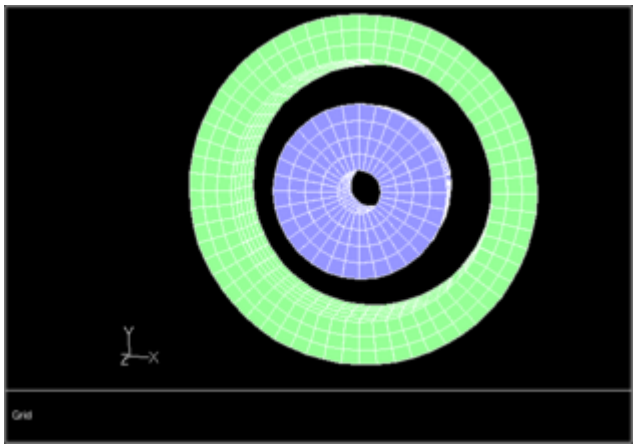


Figure 31.48: After Applying Periodicity



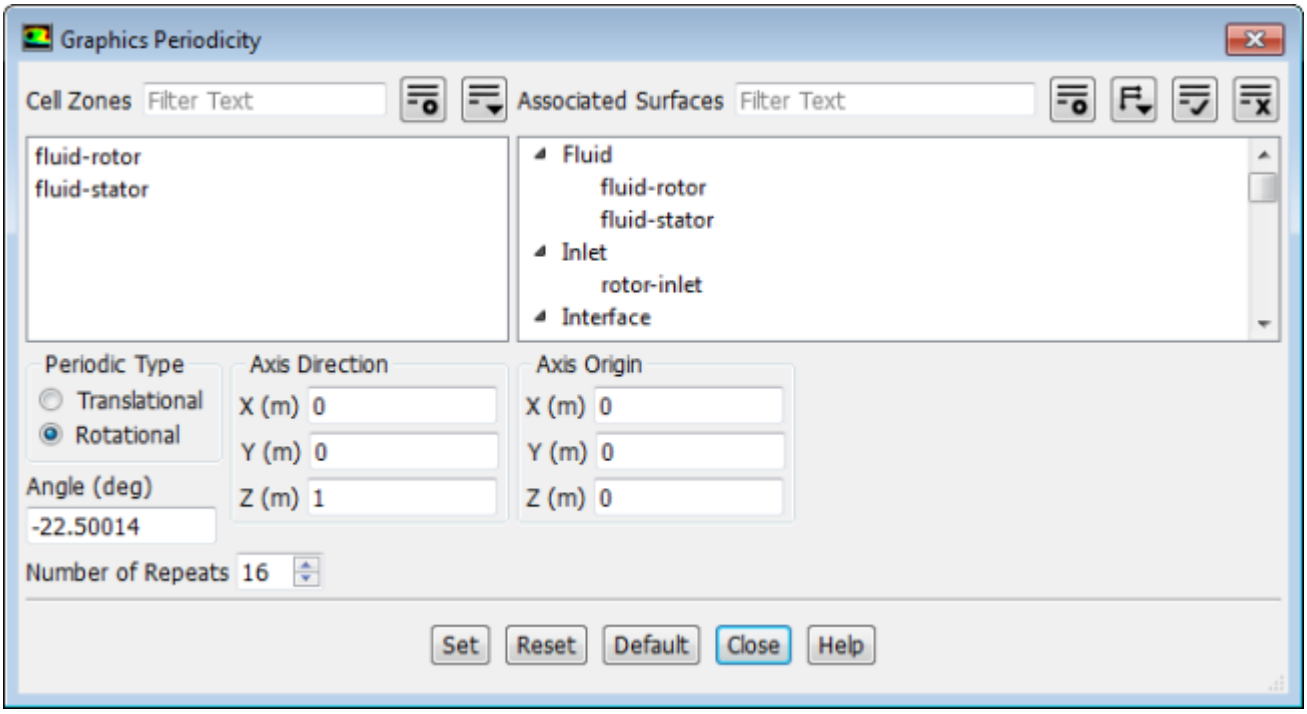
31.5.5.1. Periodic Repeats for Graphics

To define graphical periodicity for a non-periodic domain, do the following:

1. Click **Define...** under **Periodic Repeats** in the Views Dialog Box (p. 2583).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.49: The Graphics Periodicity Dialog Box



2. In the resulting Graphics Periodicity Dialog Box (p. 2586) (Figure 31.49: The Graphics Periodicity Dialog Box (p. 1814)), select the **Cell Zone** for which you want to specify the number of repeats.

   Associated Surfaces *list contains the surfaces associated with the selected cell zone. This is only informative and you cannot edit the selection of surfaces in this box.*

3. Specify **Rotational** or **Translational** as the **Periodic Type**.

4. For translational periodicity, specify the **Translation** distance of the repeated domain in the **X**, **Y**, and **Z** directions. For rotational periodicity, specify the axis about which the periodicity is defined and the **Angle** by which the domain is rotated to create the periodic repeat. For 3D problems, the axis of rotation is the vector passing through the specified **Axis Origin** and parallel to the vector from (0,0,0) to the (**X,Y,Z**) point specified under **Axis Direction**. For 2D problems, you will specify only the **Axis Origin**; the axis of rotation is the z-direction vector passing through the specified point.

5. Specify **Number of Repeats** for the selected cell zone.

6. Click **Set** in the Graphics Periodicity Dialog Box (p. 2586).

7. Follow the same procedure for other cell zones.

8. Click **Apply** in the Views Dialog Box (p. 2583) to visualize the modified display.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can delete the definition of any periodicity you have defined for graphics by clicking **Reset** in the Graphics Periodicity Dialog Box (p. 2586).

Note

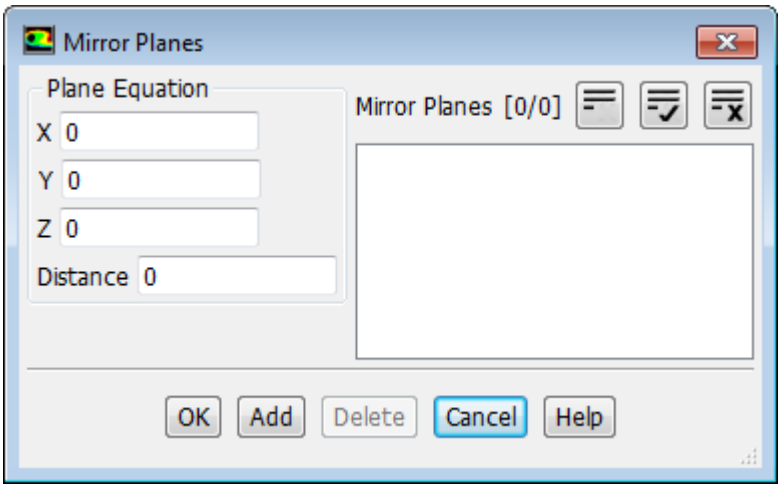
For the 3D domain with multiple periodic zones having different periodicity, ANSYS Fluent can repeat only mesh, contour and vector plots, and not the pathlines and particle tracks. Also if such domain contains, isosurfaces and clip-surfaces, that are associated with a particular cell zone, they are repeated using the same periodicity that is defined for that cell zone. However, if the surface is not associated with any cell zone, you cannot specify the periodicity for that surface.

31.5.5.2. Mirroring for Graphics

To define a mirror plane for a non-symmetric domain, do the following:

1. Click **Define Plane...** under **Mirror Planes** in the Views Dialog Box (p. 2583).

   Figure 31.50: The Mirror Planes Dialog Box

2. In the resulting Mirror Planes Dialog Box (p. 2585) (Figure 31.50: The Mirror Planes Dialog Box (p. 1815)), set the coefficients of **X**, **Y**, and **Z** and the **Distance** (of the plane from the origin) in the following equation for the mirror plane:

   $$Ax+By+Cz=\text{distance} \tag{31.1}$$

3. Click **Add** to add the defined plane to the **Mirror Planes** list. When you are done creating mirror planes, click **OK**. The newly defined plane(s) will now appear in the **Mirror Planes** list in the Views Dialog Box (p. 2583). To include the mirroring in the display, select the plane(s) and click **Apply**, as described above.

If you want to delete a mirror plane that you have defined, select it in the **Mirror Planes** list in the Mirror Planes Dialog Box (p. 2585) and click **Delete**. When you click **OK** in this dialog box, the deleted plane will be removed permanently from the **Mirror Planes** list in the Views Dialog Box (p. 2583).

31.6. Advanced Scene Composition

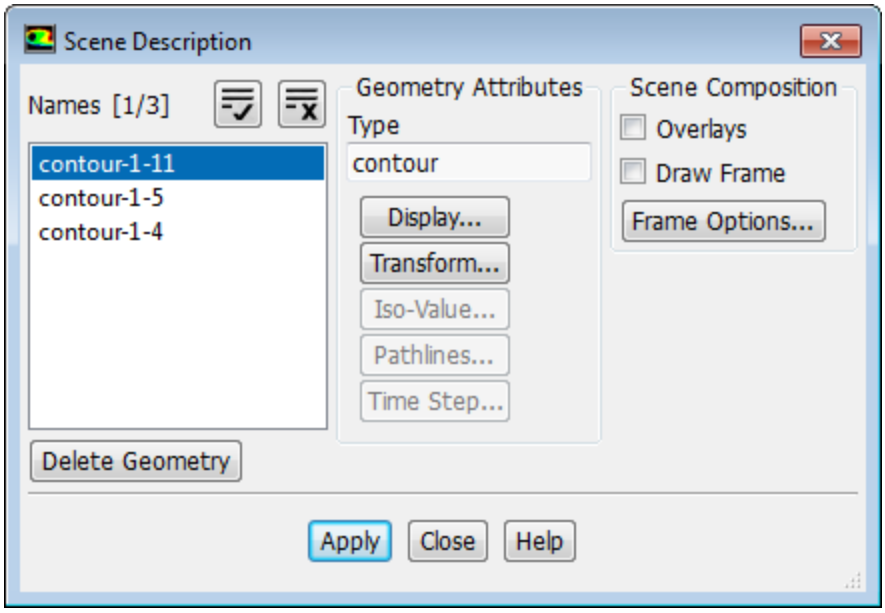
Figure 31.51: The Scene Description Dialog Box (p. 1816) allows you to create a more complex scene than what you can create using Figure 31.26: The Scene Dialog Box (p. 1781).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Once you have displayed some geometric objects (meshes, surfaces, contours, vectors, and so on) in your graphics window, you may want to move them around and change their characteristics to increase the effectiveness of the scene displayed. You can use the Scene Description Dialog Box (p. 2577) (Figure 31.51: The Scene Description Dialog Box (p. 1816)) and the Display Properties Dialog Box (p. 2578) (Figure 31.52: The Display Properties Dialog Box (p. 1817)) and Transformations Dialog Box (p. 2580) (Figure 31.54: The Transformations Dialog Box (p. 1820)), which are opened from within it, to rotate, translate, and scale each object individually, as well as change the color and visibility of each object.

Viewing → Graphics → Compose...

Figure 31.51: The Scene Description Dialog Box



The Iso-Value Dialog Box (p. 2581) (Figure 31.55: The Iso-Value Dialog Box (p. 1821)), which is also opened from within the Scene Description Dialog Box (p. 2577), enables you to change the isovalue of a selected isosurface. The Pathline Attributes Dialog Box (p. 2582) (Figure 31.56: The Pathline Attributes Dialog Box (p. 1822)) lets you set some pathline attributes. The ability to make geometric objects visible and invisible is especially useful when you are creating an animation (see Animating Graphics (p. 1825)) because it enables you to add or delete objects from the scene one at a time. The ability to change the color and position of an object independently of the others in the scene is also useful for setting up animations, as is the ability to change isosurface isovalues. You will find the features in the Scene Description Dialog Box (p. 2577) useful even when you are not generating animations because they enable you to manage your graphics window efficiently. The procedure for overlaying graphics, which uses the **Scene Description** dialog box, is described in Advanced Graphics Overlays (p. 1785). (Note that you cannot use the Scene Description Dialog Box (p. 2577) to control XY plot and histogram displays.)

For additional information, see the following sections:

31.6.1. Selecting the Object(s) to be Manipulated
31.6.2. Changing an Object's Display Properties
31.6.3. Transforming Geometric Objects in a Scene
31.6.4. Modifying Iso-Values
31.6.5. Modifying Pathline Attributes
31.6.6. Deleting an Object from the Scene
31.6.7. Adding a Bounding Frame

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.6.1. Selecting the Object(s) to be Manipulated

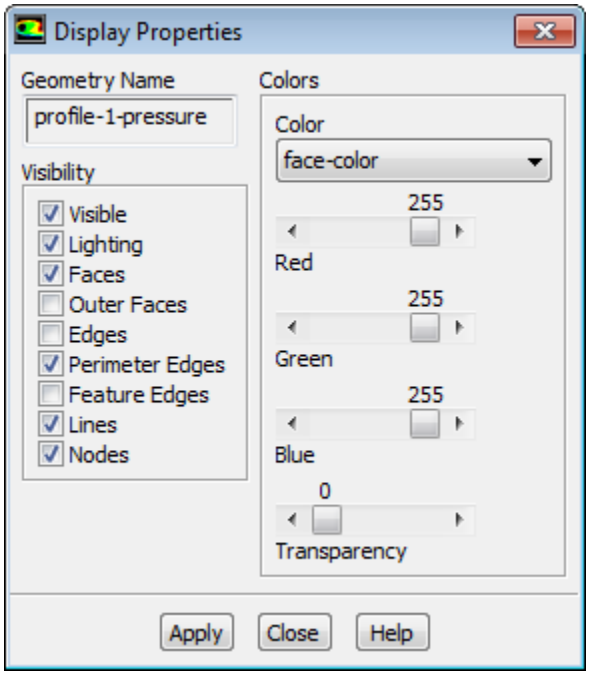
In order to manipulate the objects in the scene, you will begin by selecting the object or objects of interest in the **Names** list in the Scene Description Dialog Box (p. 2577) (Figure 31.51: The Scene Description Dialog Box (p. 1816)). The **Names** list is a list of the geometric objects that currently exist in the scene (including those that are presently invisible). If you select more than one object at a time, any operation (transformation, color specification, and so on) will apply to all the selected objects. You can also select objects by clicking on them in the graphics display using the mouse-probe button, which is, by default, the right mouse button. (See Controlling the Mouse Button Functions (p. 1800) for information about mouse button functions.) To deselect a selected object, simply click its name in the **Names** list.

When you select one or more objects (either in the **Names** list or in the display), the **Type** field will report the type of the selected object(s). Possible types for a single object include `mesh`, `surface`, `contour`, `vector`, `path`, and `text` (that is, annotation text). This information is especially helpful when you need to distinguish two or more objects with the same name. When more than one object is selected, the type displayed is `Group`.

31.6.2. Changing an Object's Display Properties

To enhance the scene in the graphics window, you can change the color, visibility, and other display properties of each geometric object in the scene. You can specify different colors for displaying the edges and faces of a mesh object to show the underlying mesh (edges) when the faces of the mesh are filled and shaded. You can also make a selected object temporarily invisible. If, for example, you are displaying the entire mesh for a complicated problem, you can make objects visible or invisible to display only certain boundary zones of the mesh without regenerating the mesh display using the Mesh Display Dialog Box (p. 2163). You can also use the visibility controls to manipulate geometric objects for efficient graphics display or for the creation of animations. These features, plus several others, are available in the Display Properties Dialog Box (p. 2578) (Figure 31.52: The Display Properties Dialog Box (p. 1817)).

Figure 31.52: The Display Properties Dialog Box



To set the display properties described above, select one or more objects in the **Names** list in the Scene Description Dialog Box (p. 2577) and then click **Display...** to open the Display Properties Dialog Box (p. 2578) for that object or group of objects.

31.6.2.1. Controlling Visibility

There are several ways for you to control the visibility of an object. All visibility options are listed under the **Visibility** heading in the Display Properties Dialog Box (p. 2578).

- To make the selected object(s) invisible, turn off the **Visible** option. To "undo" invisibility, enable the **Visible** option again.
- To turn the effect of lighting for the selected object(s) on or off, use the **Lighting** check box. You can choose to have lighting affect only certain objects instead of all of them. Note that if **Lighting** is turned on for an object such as a contour or vector plot, the colors in the plot will not be exactly the same as those in the colormap at the left of the display.
- To toggle the filled display of faces for the selected mesh or surface object(s), use the **Faces** option. Turning **Faces** on here has the same effect as turning it on for the entire mesh in the Mesh Display Dialog Box (p. 2163).
- To turn the display of outer edges on or off, use the **Outer Faces** option. This option is useful for displaying both sides of a slit wall. By default, when you display a slit wall, one side will "bleed" through to the other. When you turn off the **Outer Faces** option, the display of a slit wall will show each side distinctly as you rotate the display. This option can also be useful for displaying two-sided walls (that is, walls with fluid or solid cells on both sides).
- To turn the display of interior and exterior edges of the geometric object(s) on or off, use the **Edges** option.
- To turn the display of the outline of the geometric object(s) on or off, use the **Perimeter Edges** check box.
- To toggle the display of feature lines (described in Adding Features to an Outline Display (p. 1753)), if any, for the selected object(s), use the **Feature Edges** option.
- To toggle the display of the lines (if any) in the geometric object(s), use the **Lines** check box. Pathlines, line contours, and vectors are "lines".
- To toggle the display of nodes (if any) in the geometric object(s), use the **Nodes** check box.

Once you have set the appropriate display parameters, click **Apply** to update the graphics display.

31.6.2.2. Controlling Object Color and Transparency

The Display Properties Dialog Box (p. 2578) also lets you control an object's color and how transparent it is. All color and transparency options are listed under the **Colors** heading.

- To modify the color of faces, edges, or lines in the selected object(s), choose **face-color**, **edge-color**, **line-color**, or **node-color** in the **Color** drop-down list. The **Red**, **Green**, and **Blue** color scales will show the RGB components of the face, edge or line color, which you can modify by moving the sliders on the color scales. When you are satisfied with the color specification, click **Apply** to save it and update the display. The ability to set the colors for faces and edges can be useful when you want to have a filled display for the mesh or surface, but you also want to be able to see the mesh lines. You can achieve this effect by specifying different colors for the faces and the edges.
- To set the relative transparency of an object, select **face-color** in the **Color** drop-down list. Move the slider on the **Transparency** scale and click **Apply** to update the graphics display. An object with a transparency

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of 0 is opaque, and an object with a transparency of 100 is transparent. By specifying a high transparency value for the walls of a pipe, for example, you will be able to see contours that you have displayed on cross-sections inside the pipe. This feature is available on all platforms when the software z buffer is used for hidden surface removal, but if your display hardware supports transparency, it will be more efficient to use the hardware z buffer as the hidden surface method instead. You can select these methods in the Display Options Dialog Box (p. 2573), as described in Modifying the Rendering Options (p. 1798).

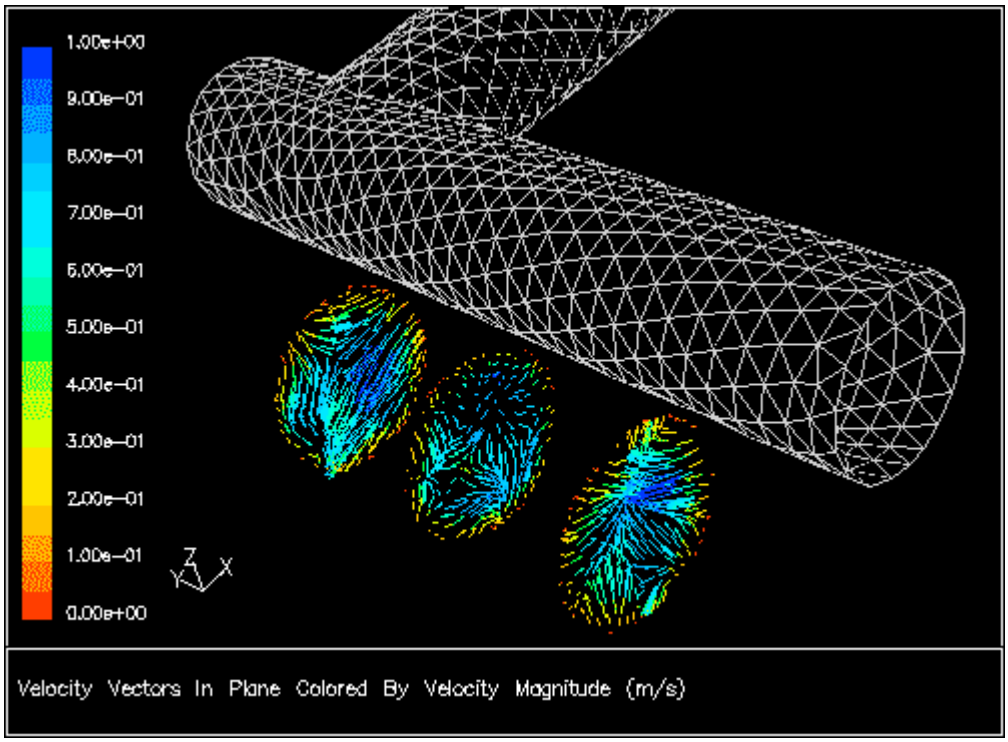
Important

If you save a picture of a display with transparent surfaces, you should *not* set the **File Type** in the Save Picture Dialog Box (p. 2568) to **Vector**.

31.6.3. Transforming Geometric Objects in a Scene

When you are composing a scene in your graphics window, you might find it helpful to move a particular object from its original position or to increase or decrease its size. For example, if you have displayed contours or vectors on cross-sections of an internal flow domain (such as a pipe), you might want to translate these cross-sections so that they will appear outside of the pipe, where they can be seen and interpreted more easily. Figure 31.53: Velocity Vectors Translated Outside the Domain for Better Viewing (p. 1819) shows such an example.

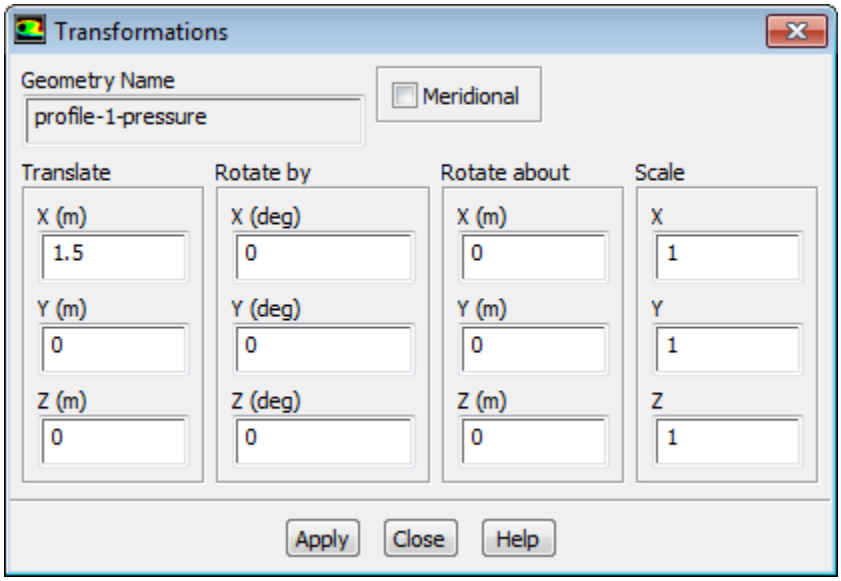
Figure 31.53: Velocity Vectors Translated Outside the Domain for Better Viewing



You can also move an object by rotating it about the x-, y-, or z-axis. If you want to display one object more prominently than the others, you can scale its size. If your geometry is rotating or has rotational symmetry, you can display the meridional view. All of these capabilities are available in the Transformations Dialog Box (p. 2580) (Figure 31.54: The Transformations Dialog Box (p. 1820)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.54: The Transformations Dialog Box



To perform the transformations described above, select one or more objects in the **Names** list in the Scene Description Dialog Box (p. 2577) and then click **Transform...** to open the Transformations Dialog Box (p. 2580) for that object or group of objects.

31.6.3.1. Translating Objects

To translate the selected object(s), enter the translation distance in each direction in the **X**, **Y**, and **Z** real number fields under **Translate**. Note that you can check the domain extents in the Scale Mesh Dialog Box (p. 2162) or the Iso-Surface Dialog Box (p. 2721). Translations are not cumulative, so you can easily return to a known state. To return to the original position, simply enter 0 in all three real number fields.

31.6.3.2. Rotating Objects

To rotate the selected object(s), enter the number of degrees by which to rotate about each axis in the **X**, **Y**, and **Z** integer number fields under **Rotate By**. You can enter any value between -360 and 360. By default, the rotation origin will be (0,0,0). If you want to spin an object about its own origin, or about some other point, specify the **X**, **Y**, and **Z** coordinates of that point under **Rotate About**.

Rotations are not cumulative, so you can easily return to a known state. To return to the original position, simply enter 0 in all three integer number fields under **Rotate By**.

31.6.3.3. Scaling Objects

To scale the selected object(s), enter the amount by which to scale in each direction in the **X**, **Y**, and **Z** real number fields under **Scale**. To avoid distortion of the object's shape, be sure to specify the same value for all three entries. Scaling is not cumulative, so you can easily return to a known state. To return the object to its original size, simply enter 1 in all three real number fields.

31.6.3.4. Displaying the Meridional View

To display the meridional view of the selected object(s), enable the **Meridional** option. This option is available only for 3D models. It is applicable to cases with a defined axis of rotation and is especially useful in turbomachinery applications.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The meridional transformation projects the selected entities onto a surface of constant angular coordinate, θ. The resultant projection thus lies in an (r, ζ) plane where ζ is in the direction of the rotation axis and r is normal to it. The value of θ used for the projection is taken as that corresponding to the minimum (r, ζ) point of the entity.

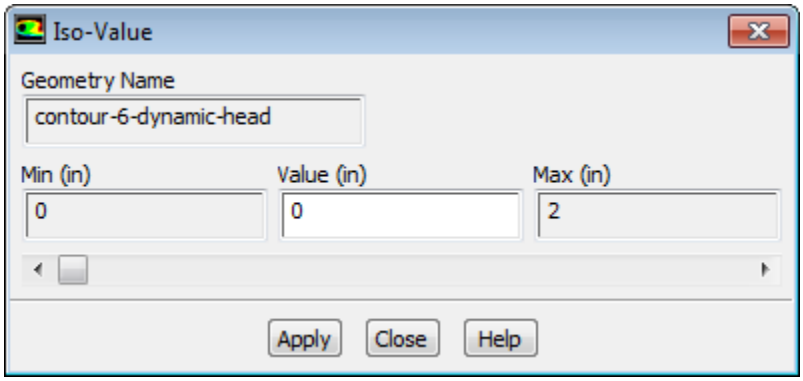
31.6.4. Modifying Iso-Values

One convenient feature that you can use to generate effective animations is the ability to generate surfaces with intermediate values between two isosurfaces with different isovalues. If the surfaces have contours, vectors, or pathlines displayed on them, ANSYS Fluent will generate and display contours, vectors, or pathlines on the intermediate surfaces that it creates.

31.6.4.1. Steps for Modifying Iso-Values

You can modify an isosurface's isovalue directly by selecting it in the **Names** list in the Scene Description Dialog Box (p. 2577) or indirectly by selecting an object displayed on the isosurface. Then click **Iso-Value...** to open the Iso-Value Dialog Box (p. 2581) (Figure 31.55: The Iso-Value Dialog Box (p. 1821)) for the selected object. Note that this button is available only if the geometric object selected in the **Names** list is an isosurface or an object on an isosurface (contour on an isosurface, for example); otherwise it is grayed out.

Figure 31.55: The Iso-Value Dialog Box



In the Iso-Value Dialog Box (p. 2581), set the new isovalue in the **Value** field, and click **Apply**. Contours, vectors, or pathlines that were displayed on the original isosurface will be displayed for the new isovalue.

31.6.4.2. An Example of Iso-Value Modification for an Animation

The ability to generate intermediate surfaces with data displayed on them is especially convenient if you want to create an animation that shows data on successive slices of the problem domain. For example, if you have solved the flow through a pipe junction and you want to create an animation that moves through one of the pipes (along the *y*-axis) and displays pressure contours on several cross-sections, you can use the following procedure:

1. Generate a surface of constant *y* coordinate (such as the *y* coordinate at the pipe inlet) using the Iso-Surface Dialog Box (p. 2721).

2. Use the Contours Dialog Box (p. 2676) to generate contours of static pressure on this isosurface and manipulate the graphics display to the desired view.

3. Open the Animate Dialog Box (p. 2567) and create key frame 1.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. In the Scene Description Dialog Box (p. 2577), select the contour in the **Names** list and click **Iso-Value...** to open the Iso-Value Dialog Box (p. 2581).

5. Change the value of the isovalue to the *y* coordinate at the other end of the pipe, and click **Apply**. You will see the contours of static pressure at the new *y* coordinate.

6. Set key frame 10 in the Animate Dialog Box (p. 2567).

7. Play back the animation.

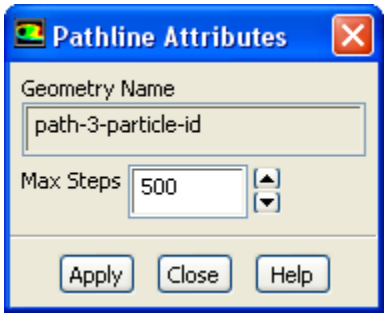
When you play back the animation, ANSYS Fluent will create the intermediate frames showing contours of static pressure on the slices between the two ends of the pipe. Ten slices will be shown in succession, all with contours displayed on them.

Using the Sweep Surface Dialog Box (p. 2565) to animate the display of contours or vectors on a surface that sweeps through the domain may be more convenient than the procedure described above. See Displaying Results on a Sweep Surface (p. 1782) for details.

31.6.5. Modifying Pathline Attributes

If you are creating animations of existing pathlines, you may want to change the number of steps used in the computation of the pathlines. This enables you to animate pathlines advancing through the domain. To do so, select the pathlines in the **Names** list in the Scene Description Dialog Box (p. 2577) and then click **Pathlines...** to open the Pathline Attributes Dialog Box (p. 2582) (Figure 31.56: The Pathline Attributes Dialog Box (p. 1822)).

Figure 31.56: The Pathline Attributes Dialog Box



In the Pathline Attributes Dialog Box (p. 2582), set the new maximum number of steps for pathline computation (**Max Steps**). After you change the value and click **Apply**, the selected pathline will be recomputed and redrawn.

31.6.5.1. An Example of Pathline Modification for an Animation

You can use the following procedure to animate pathlines from step 2 to step 101 (for example):

1. Generate the plot of pathlines using the Pathlines Dialog Box (p. 2771).

2. In the Scene Description Dialog Box (p. 2577), select the pathlines in the **Names** list and click **Pathlines...** to open the Pathline Attributes Dialog Box (p. 2582).

3. Change the value of the maximum number of steps to 2, and click **Apply**.

4. Open the Animate Dialog Box (p. 2567) and create key frame 1.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. In the Pathline Attributes Dialog Box (p. 2582), change the value of the maximum number of steps to 101, and click **Apply**.

6. Set key frame 100 in the Animate Dialog Box (p. 2567).

7. Play back the animation.

When you play back the animation, ANSYS Fluent will animate the pathlines so that they advance one step in each frame.

31.6.6. Deleting an Object from the Scene

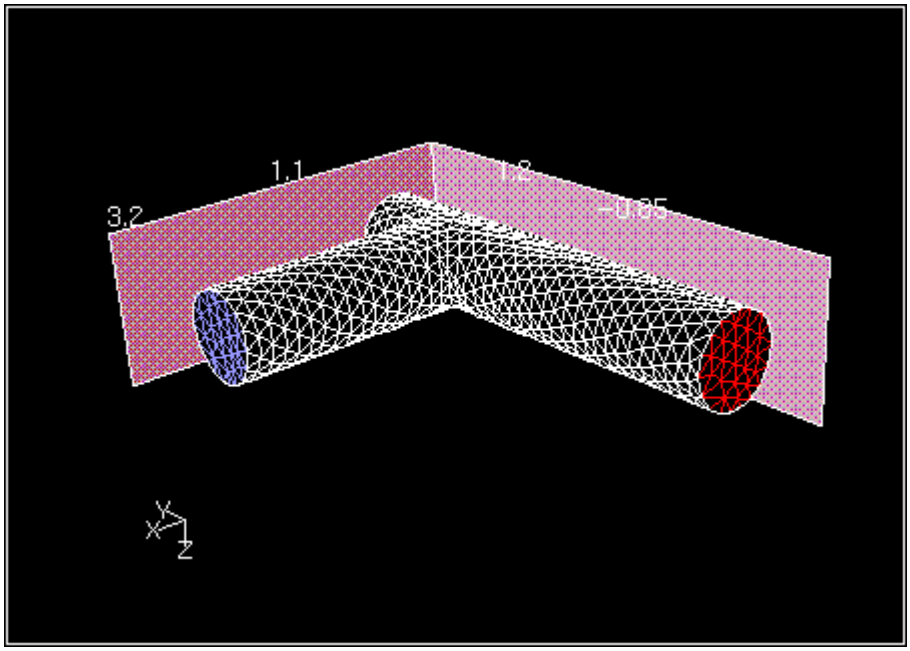
If you are composing a complex scene with overlays and find that you no longer want to keep one of the objects, it is possible to delete it without affecting any of the other objects in the scene. The ability to delete individual objects is especially useful if you have overlays on and you generate an unwanted object (for example, if you generate contours of the wrong variable). You can simply delete the unwanted object and continue your scene composition, instead of starting over from the beginning. Note that it is also possible to make objects temporarily invisible, as described in Controlling Visibility (p. 1818).

Object deletion is performed in the Scene Description Dialog Box (p. 2577) (Figure 31.51: The Scene Description Dialog Box (p. 1816)). To delete an object from the scene, select it in the **Names** list and then click **Delete Geometry**. The selected name will disappear from the **Names** list, and the display will be updated immediately.

31.6.7. Adding a Bounding Frame

ANSYS Fluent enables you to add a bounding frame around your displayed domain. You may also include measure markings on the bounding frame to indicate the length, height, and/or width of the domain, as shown in Figure 31.57: Graphics Display with Bounding Frame (p. 1823).

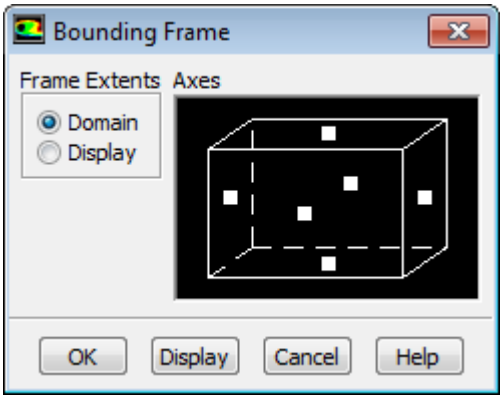
Figure 31.57: Graphics Display with Bounding Frame



To add a bounding frame to your display, you will follow the procedure below:

1. Click **Frame Options...** in the Scene Description Dialog Box (p. 2577) (Figure 31.51: The Scene Description Dialog Box (p. 1816)) to open the Bounding Frame Dialog Box (p. 2582) (Figure 31.58: The Bounding Frame Dialog Box (p. 1824)).

Figure 31.58: The Bounding Frame Dialog Box



2. Under **Frame Extents** in the Bounding Frame Dialog Box (p. 2582), select **Domain** or **Display** to indicate whether the bounding frame should encompass the domain extents or only the portion of the domain that is shown in the display.

3. In the **Axes** portion of the Bounding Frame Dialog Box (p. 2582), specify the frame boundaries and measurements to be shown in the display:

   - Indicate the bounding plane(s) (for example, the *x*-*z* and *y*-*z* planes shown in Figure 31.57: Graphics Display with Bounding Frame (p. 1823)) to be displayed by clicking on the white square on the appropriate plane of the box shown under the **Axes** heading. You can use any of the mouse buttons. The square will turn red to indicate that the associated bounding plane will be displayed in the graphics window.

   - Specify where you would like to see the measurement annotations by clicking on the appropriate edge of the box. The edge will turn red to indicate that the markings will be displayed along that edge of the displayed geometry.

   Important

   If you have trouble determining which square or edge corresponds to which location in your domain, you can easily find out by displaying one or two bounding planes to get your bearings. You can then select the appropriate objects to obtain the final display.

4. Click **Display** to update the display with the current settings. If you are not satisfied with the frame, repeat steps 2 and/or 3 and click **Display** again.

5. Once you are satisfied with the bounding frame that is displayed, click **OK** to close the Bounding Frame Dialog Box (p. 2582) and save the frame settings for future displays.

6. If you want to include the bounding frame in all subsequent displays, enable the **Draw Frame** option in the Scene Description Dialog Box (p. 2577) and click **Apply**. If this option is not enabled, the bounding box will appear only in the current display; it will not be redisplayed when you generate a new display (unless you have overlays enabled).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

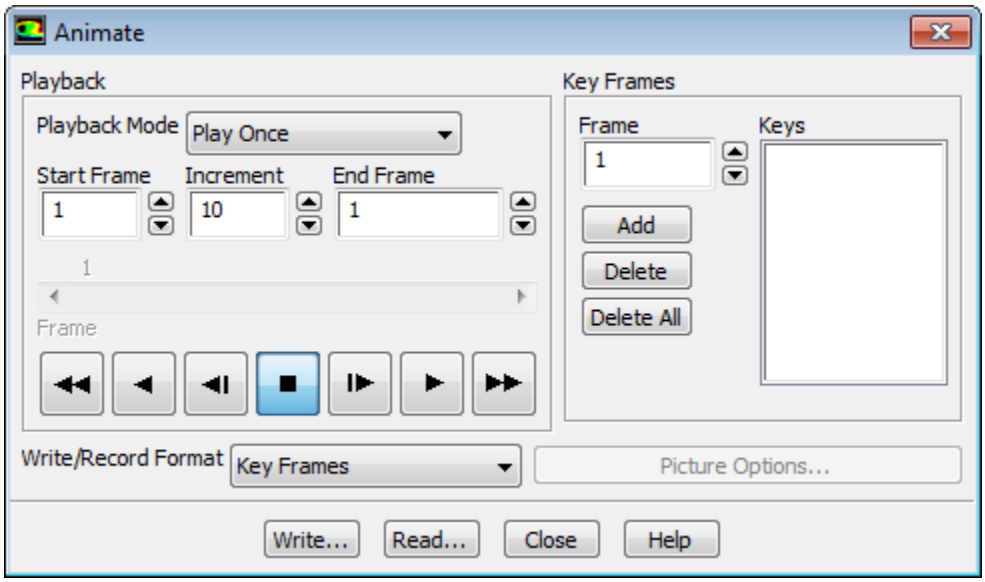
The bounding planes and axis annotations will appear in the **Names** list of the Scene Description Dialog Box (p. 2577), and you can manipulate them in the same way as any other geometric object in the display. For example, you can use the Display Properties Dialog Box (p. 2578) to change the face color of a bounding plane or to make it transparent (see Changing an Object's Display Properties (p. 1817)).

31.7. Animating Graphics

To generate animations that progress from one static view of the graphics display to the next, you can set up "key frames" (individual static images) using the Animate Dialog Box (p. 2567) (Figure 31.59: The Animate Dialog Box (p. 1825)).

Results → Animations → Scene Animation → Edit...

Figure 31.59: The Animate Dialog Box



You can compose a scene in the graphics window and define it as a single key frame. Then, modify the scene by moving or scaling objects, making some objects invisible or visible, changing colors, changing the view, or making other changes, and define the new scene as another key frame. ANSYS Fluent can then interpolate smoothly between the two frames that you defined, creating a specified number of intermediate frames.

Another method of generating animations is to automatically generate surfaces with intermediate values between two isosurfaces with different isovalues. See Modifying Iso-Values (p. 1821) for details. See Displaying Results on a Sweep Surface (p. 1782) for information about displaying the mesh, contours, or vectors on a surface that sweeps through the domain. If you want to create a graphical animation of the solution over time, you can use Figure 28.43: The Animation Definition Dialog Box (p. 1648) to set up the graphical displays that you want to use in the animation. You can choose the type of display you want to animate by selecting it from the **Animation Object** list or by creating a new animation object. For details on animating the solution, see Animating the Solution (p. 1647). For more information on generating, displaying, and saving pathlines and particle tracks, refer to Displaying Pathlines (p. 1772).

For additional information, see the following sections:

31.7.1. Creating an Animation
31.7.2. Playing an Animation
31.7.3. Saving an Animation

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.7.4. Reading an Animation File
31.7.5. Notes on Animation

31.7.1. Creating an Animation

You can define any number of key frames (up to 3000) to create your animation. By assigning the appropriate numbers to the key frames, you provide the information ANSYS Fluent needs to create the correct number of intermediate frames. For example, to create a simple animation that begins with a front view of an object, moves to a side view, and ends with a rear view of the object, you would follow the procedure outlined below:

1. Determine the number of frames that you want in the animation. For this example, consider the animation to be 31 frames.

2. Determine the number of key frames that you need to specify. In this example, you will specify three: one showing the front view, one showing the side view, and one showing the rear view.

3. Determine the appropriate key frame numbers to assign to the 3 specified frames. Here, the front view will be specified as key frame 1, the side view will be key frame 16, and the rear view will be key frame 31.

4. Compose the scenes for each view to be used as a key frame. You can use the Scene Description Dialog Box (p. 2577) (see Advanced Scene Composition (p. 1815)) and the Views Dialog Box (p. 2583) (see Modifying the View (p. 1803)) to modify the display, and any other dialog boxes or commands to create contours, vectors, pathlines, and so on to be included in each scene. After you complete each scene, create the appropriate key frame by setting the **Frame** number and clicking **Add** under **Key Frames** in the Animate Dialog Box (p. 2567). For special considerations related to key frame definition, see Notes on Animation (p. 1829).

   Important

   Be sure to change the **Frame** number before clicking **Add**, or you will overwrite the last key frame that you created.

   You can check any of the key frames that you have created by selecting it in the **Keys** list. The selected key frame will be displayed in the graphics window.

5. When you complete the animation, you can play it back as described in Playing an Animation (p. 1826) and/or save it as described in Saving an Animation (p. 1828).

31.7.1.1. Deleting Key Frames

If, during the creation of your animation, you want to delete one of the key frames that you have defined, select the key frame in the **Keys** list and click **Delete**. If you want to delete all key frames and start over again, click **Delete All**.

31.7.2. Playing an Animation

Once you have defined the key frames (as described in Creating an Animation (p. 1826)) or read in a previously created animation file (as described in Reading an Animation File (p. 1829)), you can play back the animation and ANSYS Fluent will interpolate between the frames that you specified to complete the animation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To play the animation once through from start to finish, click the “play” button under the **Playback** heading in the Animate Dialog Box (p. 2567). (The buttons function in a way similar to those on a standard video cassette player. “Play” is the second button from the right—a single triangle pointing to the right.) To play the animation backwards once, click the “play reverse” button (the second from the left—a single triangle point to the left). As the animation plays, the **Frame** scale shows the number of the frame that is currently displayed, as well as its relative position in the entire animation. If, instead of playing the complete animation, you want to jump to a particular key frame, move the **Frame** slider bar to the desired frame number, and the frame corresponding to the new frame number will be displayed in the graphics window.

Additional options for playing back animations are described below. Be sure to check Notes on Animation (p. 1829) as well for important notes about playing back animations.

31.7.2.1. Playing Back an Excerpt

You may sometimes want to play only one portion of a long animation. To do this, you can modify the **Start Frame** and the **End Frame** under the **Playback** heading in the Animate Dialog Box (p. 2567). For example, if your animation contains 50 frames, but you want to play only frames 20 to 35, you can set **Start Frame** to 20 and **End Frame** to 35. When you play the animation, it will start at frame 20 and finish at frame 35.

31.7.2.2. "Fast-Forwarding" the Animation

You can "fast-forward" or "fast-reverse" the animation by skipping some of the frames during playback. To fast-forward the animation, set the **Increment** and click the fast-forward button. If, for example, your **Start Frame** is 1, your **End Frame** is 15, and your **Increment** is 2, when you click the fast-forward button (the last button on the right—two triangles pointing to the right), the animation will show frames 1, 3, 5, 7, 9, 11, 13 and 15. Clicking on the fast-reverse button (the first button on the left—two triangles pointing to the left) will show frames 15, 13, 11,...1.

31.7.2.3. Continuous Animation

If you want the playback of the animation to repeat continuously, there are two options available.

- To continuously play the animation from beginning to end (or from end to beginning, if you use one of the reverse play buttons), select **Auto Repeat** in the **Playback Mode** drop-down list in the Animate Dialog Box (p. 2567).
- To play the animation back and forth continuously, reversing the playback direction each time, select **Auto Reverse** in the **Playback Mode** drop-down list.

To turn off the continuous playback, select **Play Once** in the **Playback Mode** list. This is the default setting.

31.7.2.4. Stopping the Animation

To stop the animation during playback, click the "stop" button (the square in the middle of the playback control buttons). If your animation contains very complicated scenes, there may be a slight delay before the animation stops.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.7.2.5. Advancing the Animation Frame by Frame

To advance the animation manually frame by frame, use the third button from the right (a vertical bar with a triangle pointing to the right). Each time you click this button, the next frame will be displayed in the graphics window. To reverse the animation frame by frame, use the third button from the left (a left-pointing triangle with a vertical bar). Frame-by-frame playback enables you to freeze the animation at points that are of particular interest.

31.7.3. Saving an Animation

Once you have created your animation, you can save it in any of the following formats:

- Animation file containing the key frame descriptions
- Picture files, each containing a frame of the animation
- MPEG file containing each frame of the animation
- Video (see Creating Videos (p. 1829))

31.7.3.1. Animation File

You can save the key frame definitions to a file that can be read back into ANSYS Fluent (see Reading an Animation File (p. 1829)) when you want to replay the animation. Since the animation file will contain only the key frame definitions, you must be sure that you have a case and data file containing the necessary surfaces and other information referred to by the key frame descriptions.

To write an animation file, select **Key Frames** in the **Write/Record Format** drop-down list in the Animate Dialog Box (p. 2567), and click **Write...**. In The Select File Dialog Box (p. 19), specify the name of the file and save it.

31.7.3.2. Picture File

You can also generate a picture file for each frame in the animation. This feature enables you to save your animation frames to picture files used by an external animation program such as ImageMagick. To save the animation as a picture file, follow these steps:

1. Select **Picture Files** in the **Write/Record Format** drop-down list in the Animate Dialog Box (p. 2567).
2. If necessary, click **Picture Options...** to open the Save Picture Dialog Box (p. 2568) and set the appropriate parameters for saving the picture files. (If you are saving picture files for use with ImageMagick, for example, you may want to select the window dump format. See Window Dumps (Linux Systems Only) (p. 106) for details.) Click **Apply** in the Save Picture Dialog Box (p. 2568) to save your modified settings.

   Important

   Do not click **Save...** in the Save Picture Dialog Box (p. 2568). You will save the picture files from the Animate Dialog Box (p. 2567) in the next step.

3. In the Animate Dialog Box (p. 2567), click **Write...**. In The Select File Dialog Box (p. 19), specify the filename and click **OK** to save the files. (See Window Dumps (Linux Systems Only) (p. 106) for information about

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

specifying filenames that increment automatically as additional pictures are saved.) ANSYS Fluent will replay the animation, saving each frame to a separate file.

31.7.3.3. MPEG File

It is also possible to save all of the frames of the animation in an MPEG file, which can be viewed using an MPEG decoder such as **mpeg_play**. Saving the entire animation to an MPEG file will require less disk space than storing the individual window dump files (using the picture method), but the MPEG file will yield lower-quality images. To save the animation to an MPEG file, follow these steps:

1. Select **MPEG** in the **Write/Record Format** drop-down list in the Animate Dialog Box (p. 2567).

2. In the Animate Dialog Box (p. 2567), click **Write...**. In The Select File Dialog Box (p. 19), specify the filename, and click **OK** to save the files.

   ANSYS Fluent replays the animation and saves each frame to a separate scratch file; it then combines all of the files into a single MPEG file.

31.7.4. Reading an Animation File

If you have saved the key frames defining an animation to an animation file (as described in Saving an Animation (p. 1828)), you can read that file back in at a later time (or in different session) and play the animation. Before reading in an animation file, be sure that the current case and data contain the surfaces and any other information that the key frame description refers to.

To read an animation file, click **Read...** in the Animate Dialog Box (p. 2567). In The Select File Dialog Box (p. 19), specify the name of the file to be read.

31.7.5. Notes on Animation

When you are creating and playing back animations, note the following:

- For smoother animations, enable the **Double Buffering** option in the Display Options Dialog Box (p. 2573) (see Modifying the Rendering Options (p. 1798)). This will reduce screen flicker during graphics updates.

- When you are defining key frames, you must create all geometric objects that will be used in the animation before you create any key frames. You cannot create a key frame using one set of geometric objects and then generate a new geometry (such as a vector plot) and include that in another key frame. Create all geometric objects first, and then use the Display Properties Dialog Box (p. 2578) to control the visibility of the objects in each key frame (see Controlling Visibility (p. 1818)).

- A single animation sequence can contain up to 3000 key frames.

- When you play back an animation, the colormap used will be the one that is currently active, *not* the one that was active during "recording."

31.8. Creating Videos

Tools are available for creating videos from ANSYS Fluent. This section is a guide to video creation using the new video capabilities. It assumes that you have a ready-to-use video system, and that you are familiar with this system, including the special video hardware and software installed on your computer. The main use for this feature is to record an animation that you have created using the Animate Dialog Box (p. 2567) (as described in Animating Graphics (p. 1825)). This section will describe issues involved in

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

recording animations to video, the kind of video equipment you will need, and the procedures for creating a video using ANSYS Fluent.

Important

Video creation is not currently available in Windows versions of ANSYS Fluent.

For additional information, see the following sections:

31.8.1. Recording Animations To Video
31.8.2. Equipment Required
31.8.3. Recording an Animation with ANSYS Fluent

31.8.1. Recording Animations To Video

Recording an animation involves copying the computer-generated images to videotape so that you can view the animation with a VCR, or another type of tape player. This task is not an easy one, as there are several issues that should be addressed in order to create an acceptable video. A couple of these issues are described in the following sections.

31.8.1.1. Computer Image vs. Video Image

The computer monitor uses a different video signal than the video tape recorder (VTR). Most computers use an RGB-component, non-interlaced signal with high resolution and a high refresh rate. A VTR typically uses a standard broadcast video signal (such as NTSC or PAL), which has an interlaced, composite signal with lower resolution and a lower refresh rate. In order to send the computer image to the VTR for recording, the computer has to produce a video signal in the proper format. This requires extra hardware, which, in many cases, converts RGB component video to standard broadcast video, resulting in a lower quality image. A solution to this problem is to make sure that the image you are recording does not have small text, or too much small detail that will be hard to see on video. Sometimes it is best to zoom in on an area of interest in a large image and animate just that portion.

Another problem is that RGB-component video has a larger color space (or color gamut) than standard broadcast video. This means that some colors may get "clipped" when an image is converted to broadcast video, resulting in washed-out colors, or color bleeding. The solution is to try to make sure that the colors fall within the color space of the video format, and are not oversaturated. Some picture controls that can help you do this are available in ANSYS Fluent. These controls will be discussed in Check the Picture Quality (p. 1836).

31.8.1.2. Real-Time vs. Frame-By-Frame

If the images in the animation can be rendered fast enough on the computer screen, it may be possible to record the animation in real-time. This is as simple as placing the video tape recorder (VTR) in record mode, and playing the animation on the computer screen. This also requires scan-converting hardware that will convert the scan lines of the computer screen to a video signal sent to the VTR.

In many cases, however, the animation cannot be played back on the computer screen in real-time. To create a video that plays the animation at a desirable speed, the animation must be recorded frame-by-frame. This involves sending one frame to the VTR, instructing it to record the frame at a specific point on the tape, then backing up the VTR to repeat the procedure with the next frame. This process takes quite a bit longer than real-time recording, but the result can be a much smoother video animation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.8.2. Equipment Required

In general, recording an animation to video requires a system with the following hardware components:

Computer
with video hardware to produce the video signal.

Editing VTR
(video tape recorder) that supports frame-accurate recording.

VTR Controller
which enables computer software to control the recording process.

Two VTR controller models are supported by ANSYS Fluent: the V-LAN controller developed by Videomedia, Inc., and the MiniVAS/MiniVAS-2 controller developed by the V.A.S. Group. ANSYS Fluent assumes that your recording system is set up as shown in Figure 31.60: Recording System with V-LAN Controller (p. 1831) for a system with a V-LAN controller or as shown in Figure 31.61: Recording System with MiniVAS/MiniVAS-2 Controller (p. 1831) for a system with a MiniVAS controller.

Figure 31.60: Recording System with V-LAN Controller

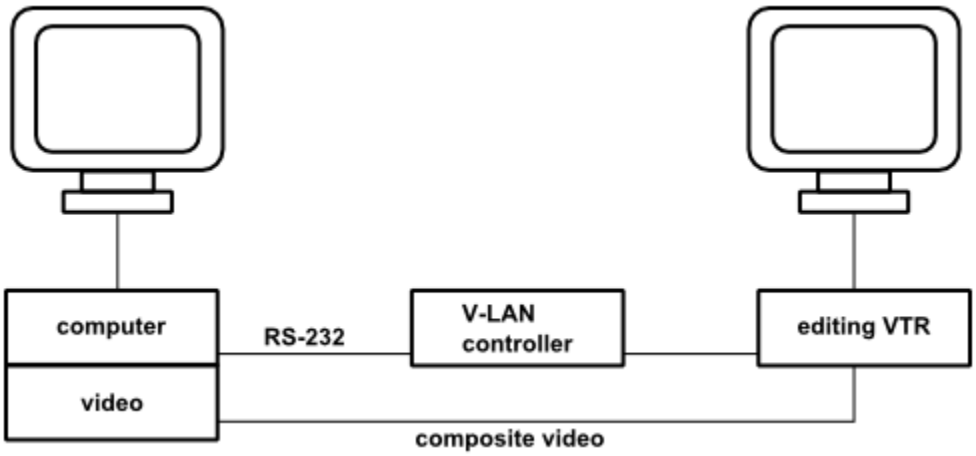
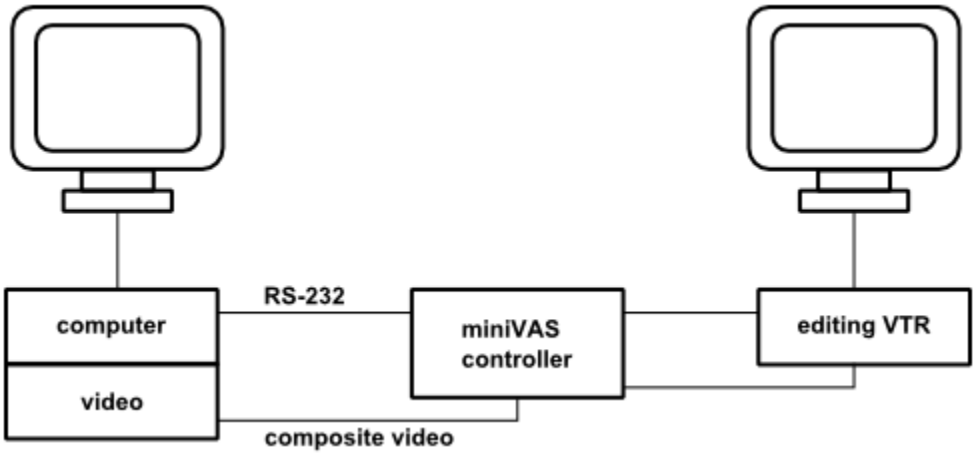


Figure 31.61: Recording System with MiniVAS/MiniVAS-2 Controller



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.8.3. Recording an Animation with ANSYS Fluent

The procedure for recording an animation using ANSYS Fluent is as follows:

- Create an Animation (p. 1832)
- Open a Connection to the VTR Controller (p. 1833)
- Set Up Your Recording Session (p. 1833)
- Check the Picture Quality (p. 1836)
- Make Sure Your Tape is Formatted (Preblacked) (p. 1837)
- Start the Recording Session (p. 1837)

Each step is described in detail in the following sections.

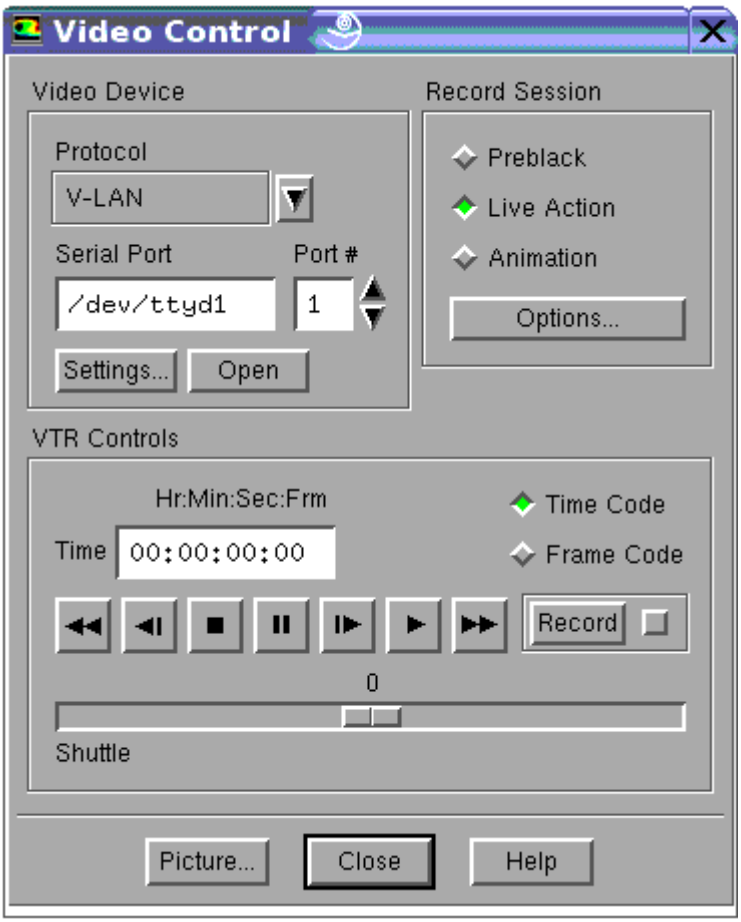
31.8.3.1. Create an Animation

When recording animations to video, you must first create your animation. It's also a good idea to play it back a couple times to make sure you are satisfied with it, and to save the animation key frame definitions to a file for later use (see Creating an Animation (p. 1826)).

When you are ready to record the animation, you can select **Video** in the **Write/Record Format** drop-down list found in the Animate Dialog Box (p. 2567). When you do so, the name of the **Write...** button will change to **Record...**, and you can click **Record...** to display the Video Control Dialog Box (p. 2833) (Figure 31.62: The Video Control Dialog Box (p. 1833)) used for video creation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.62: The Video Control Dialog Box



31.8.3.2. Open a Connection to the VTR Controller

The procedure for connecting to your VTR controller is as follows:

1. Select the protocol used by your VTR controller using the **Protocol** drop-down list.

2. Check the settings for your VTR controller by clicking on the **Settings...** button. For V-LAN, this will display the V-LAN Settings Dialog Box (p. 2836), and for MiniVAS, it will display the MiniVAS Settings Dialog Box (p. 2742).

3. Select the RS-232 serial port used to connect the VTR controller to your computer. Usually, the serial port is identified by a file name such as `/dev/ttyd1` for serial port 1, and `/dev/ttyd2` for serial port 2. If this is the case on your system, you can simply set the value of **Port #**; otherwise, you can type a new file name in the **Serial Port** text entry. Make sure that you have the proper Linux read/write permissions for the file.

4. Open a connection to the VTR controller by clicking the **Open** button. If successful, a line will be printed out in the console window that reports the VTR controller protocol version and the VTR device ID.

31.8.3.3. Set Up Your Recording Session

Once you have established a connection to the VTR controller, you can set up your recording session. There are three types of recording sessions, as described below:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Preblack
is the process of formatting a tape by laying down a time code onto the tape. A tape must be formatted before any frame-accurate editing, including frame-by-frame animation, can be performed. During this process, one usually records a black video signal onto the tape as well, therefore the name "preblack". When you select this option, the current graphics window will be cleared to black. You can use the window to send your black video signal to the VTR.

Important

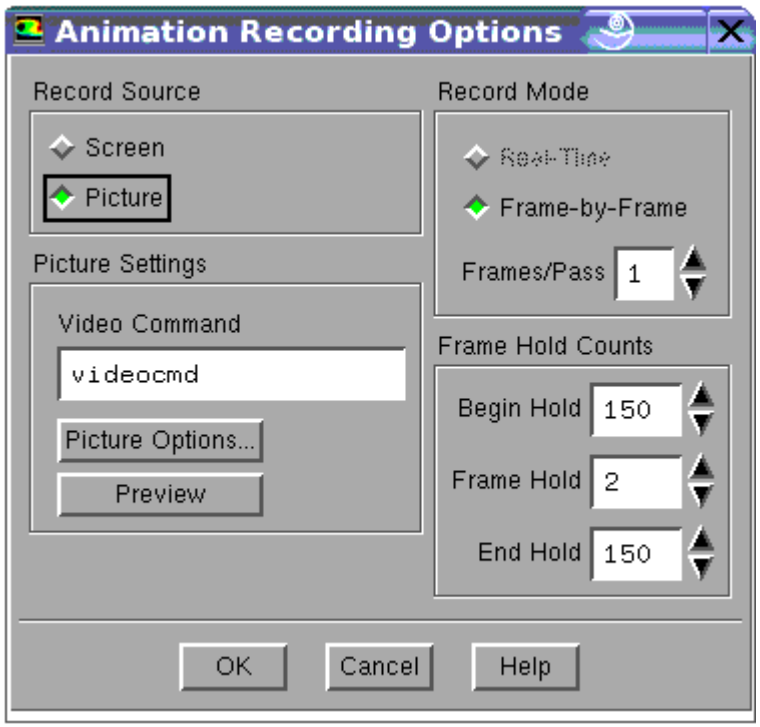
When you preblack a previously formatted tape, a new time code will be written and any previously recorded video will be destroyed.

Live Action
Enables you to record a live ANSYS Fluent session, which can be used for demonstration. This option requires your computer's video hardware to have a scan converter that will send the computer display image to your VTR system.

Animation
Plays an animation that you have created, and record it onto your VTR system.

The **Options...** button in the Video Control Dialog Box (p. 2833) is used to display the Animation Recording Options Dialog Box (p. 2668) (Figure 31.63: The Animation Recording Options Dialog Box (p. 1834)):

Figure 31.63: The Animation Recording Options Dialog Box



There are three parts to setting up your animation recording session:

1. Select the recording source.

2. Choose real-time or frame-by-frame recording.

3. Set the video frame hold counts.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.8.3.3.1. Select the Recording Source

There are two possible video sources that can be used for recording an animation: **Screen** and **Picture**. The choice of video source depends on what your video hardware/software provides. Here is a description of each:

Screen
can be used if your computer's video hardware can send all or a portion of the computer screen as a video signal to the VTR using a scan converter and associated software. With this option, you are responsible for setting up the scan converter and sending the video signal to the VTR.

Picture
instructs ANSYS Fluent to create a picture of each frame of animation and send the picture file to the computer's video hardware using a system command. This option assumes that your computer's video system includes a frame buffer that can store an image and send it as a video signal to the video recording system.

When using the picture option, a shell script will be called that will send the picture file to the video frame buffer. The default setting is `videocmd`, which is a shell script that is included in your ANSYS Fluent distribution. It is located in *path* `/ansys_inc/v181/fluent/bin`, where *path* is the folder in which you have placed the release folder. This shell script will execute your system's command to send an image file to the video frame buffer. The script `videocmd` is set up to call the SGI system command `memtovid`. If you have a different system, you must copy the shell script `videocmd` to a new file and modify it to perform the proper task on your system (see the comments in `videocmd` for details). You can specify the name of your shell script using the **Video Command** text entry in the Animation Recording Options Dialog Box (p. 2668).

In order to send a picture file of the proper format to the video frame buffer, you must set up the picture format using the Save Picture Dialog Box (p. 2568), which can be displayed by clicking the **Picture Options...** button in the Animation Recording Options Dialog Box (p. 2668). If you choose to perform a window dump to create the picture file, the default window dump command used will also be `videocmd`. You can change this setting to use your own command. After setting the picture options, click **Apply** instead of **Save...** in the Save Picture Dialog Box (p. 2568) to apply the change.

Once you have set up the picture format and system command, you can test the configuration by sending the picture in the current graphics window to the video frame buffer. This is done by clicking on **Preview** in the Animation Recording Options Dialog Box (p. 2668). Note that this is another way to send a black video signal to your VTR when you are preblacking a tape.

31.8.3.3.2. Choose Real-Time or Frame-By-Frame Recording

There are two methods for recording an animation: real-time and frame-by-frame.

Real-Time
can be used if the animation playback speed is fast enough to provide a reasonably smooth animation in real-time. This is only available if the selected record source is **Screen**. In this mode, ANSYS Fluent will simply turn VTR recording on, play the animation, then stop the recording.

Frame-By-Frame
is used to produce a higher-quality video animation by recording one frame at a time. For each animation frame, this method will 1) play the frame on the screen (and generate the picture file, if needed), 2) preroll the VTR, and 3) record the frame. If the animation has 50 frames, this procedure is repeated 50 times, that is, 50 record passes are made. This is the recommended method, because the real-time playback of the animation will usually be too slow and choppy.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When recording in frame-by-frame mode, there is an optional setting called **Frames/Pass**, which can be used to try and speed up the frame-by-frame recording process. It specifies the number of animation frames recorded to tape per record pass. If the animation is long enough (200 frames or more), you can try setting this value to 2 or higher. For example, if you set this value to 2 for a 202-frame animation, it will record animation frame 1 during the first pass, frames 2 and 102 during the second pass, frames 3 and 103 during the third pass, and so on. This is possible only if the animation frames can be rendered in time to be inserted onto the tape during a record pass, so use this setting with caution.

31.8.3.3.3. Set the Video Frame Hold Counts

The video standard NTSC has a frame rate of 30 frames/sec (and the PAL standard has a rate of 25 frames/sec). At the NTSC rate, a 150-frame animation will take only 5 seconds to play. To stretch out the animation, you can record the same animation frame over 2 or more video frames. This is done by setting video frame hold counts for the beginning, middle, and end of the animation, using the Animation Recording Options Dialog Box (p. 2668) controls.

Begin Hold
specifies the number of video frames to hold the first animation frame. It helps to hold the first frame for about 5 seconds (150 video frames for NTSC, or 125 for PAL) so that the viewer can get accustomed to the picture before the animation begins.

Frame Hold
specifies the number of video frames to hold each animation frame, other than the first and last. To slow down your recorded animation, try setting this value to 2 or 3.

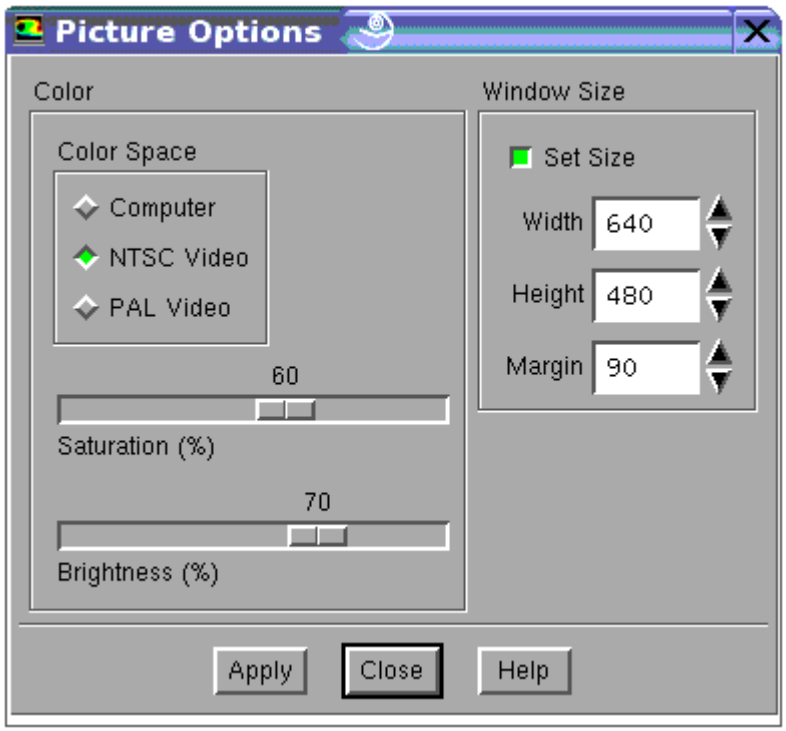
End Hold
specifies the number of video frames to hold the last animation frame. You may want to hold the last animation frame for about 5 seconds to provide closure.

31.8.3.4. Check the Picture Quality

As described in Recording Animations To Video (p. 1830), there are several sacrifices made when sending a computer image to video, including loss of color and resolution. Some steps can be taken to minimize the problem using the Picture Options Dialog Box (p. 2776) (Figure 31.64: The Picture Options Dialog Box (p. 1837)). Display this dialog box by clicking the **Picture...** button in the Video Control Dialog Box (p. 2833).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.64: The Picture Options Dialog Box



Color

Use these controls to ensure that all colors fall into the proper color space for your video device. Also, for best results, set the saturation and brightness levels to 80% or less.

Window Size

If you have a scan converter that converts a portion of the computer screen, you can set the graphics window to a particular pixel size to match the scan converter's window size. You can also create a margin around the picture in the window to keep unwanted parts of the screen (such as the window border) out of the video image.

31.8.3.5. Make Sure Your Tape is Formatted (Preblacked)

Before you can start the recording session, you need to make sure the tape has been preblacked with a time code or frame code. When you start with a brand new tape, you need to take time out and preblack the whole tape first. This can be done using the following procedure:

1. Rewind the tape to the beginning.
2. Select a preblack recording session by selecting **Preblack** in the **Record Session** group box of the Video Control Dialog Box (p. 2833).
3. Send the VTR a black video signal using a scan converter or a picture by clicking the **Preview** button in the Animation Recording Options Dialog Box (p. 2668).
4. Click **Preblack** in the **VTR Controls** section of the Video Control Dialog Box (p. 2833) to start the preblacking.

31.8.3.6. Start the Recording Session

Make sure you have the proper recording session selected. If you are recording an animation, **Animation** should be selected.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To start recording onto tape, you must first go to the "in point" on tape where you want the recording to begin. With a blank tape, it is important to start at about 20 seconds into the tape, so the VTR has a chance to preroll up to the in point. You can use the VTR button controls to position the tape, but an easier way to go to a certain point is to type the time code or frame code in the **Time** or **Frame** counter and press **Enter**. For example, a time code of `00:02:36:07` is 2 minutes, 36 seconds, and 7 frames. In order to go to this position on the tape, you can enter the time code as `2:36:07`, leaving out the leading zeros, or you can simply enter `23607`, leaving out the leading zeros and colons.

Once your tape is at the start position for your recording session, click **Record** to start recording.

31.9. Histogram and XY Plots

In addition to the many graphics tools already discussed, ANSYS Fluent also provides tools that enable you to generate XY plots and histograms of solution, file, profile, and residual data. You can modify the colors, titles, legend, and axis and curve attributes to customize your plots. The following sections describe the XY and histogram plotting features in ANSYS Fluent.

31.9.1. Plot Types
31.9.2. XY Plots of Solution Data
31.9.3. XY Plots of File Data
31.9.4. XY Plots of Profiles
31.9.5. XY Plots of Circumferential Averages
31.9.6. XY Plot File Format
31.9.7. Residual Plots
31.9.8. Histograms
31.9.9. Modifying Axis Attributes
31.9.10. Modifying Curve Attributes

31.9.1. Plot Types

Data can be plotted in XY (abscissa/ordinate) form or histogram form. Each form is described below.

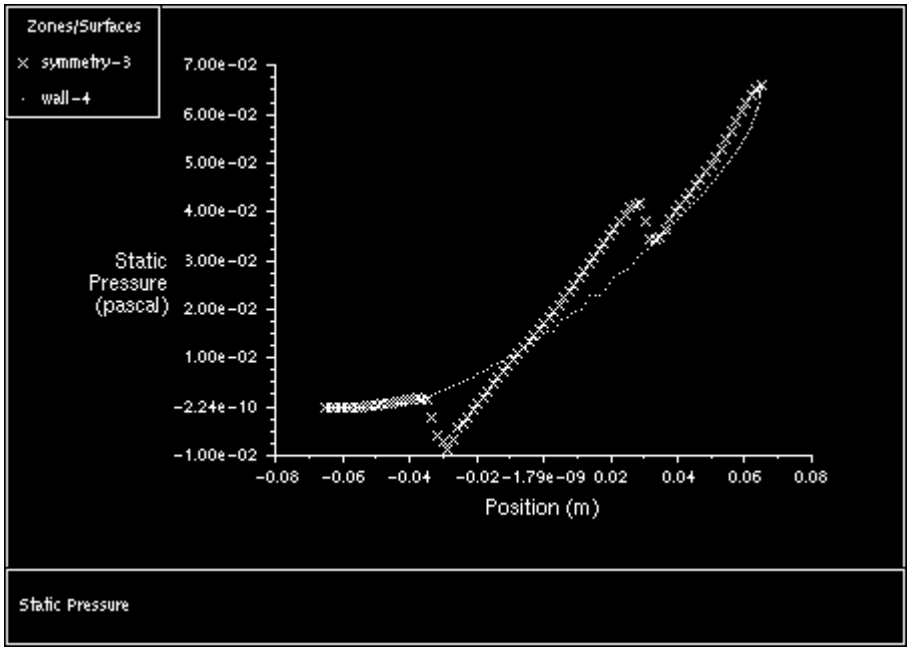
31.9.1.1. XY Plots

An XY (abscissa/ordinate) plot is a line and/or symbol chart of data. Virtually any defined variable or function is accessible for this type of plot. Furthermore, you may read in an externally-generated data file in order to compare your results with experimental data. You can also use the XY-plot facility to plot out profile data, the residual histories of variables, or the time histories if you have a transient problem.

ANSYS Fluent provides tools for controlling many aspects of the XY plot, including background color, legend, and axis and curve attributes. Figure 31.65: Sample XY Plot (p. 1839) shows a sample XY plot.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.65: Sample XY Plot



To differentiate the data being displayed, you can customize the pattern, color and weight of the data lines and the shape, color, and size of the data markers.

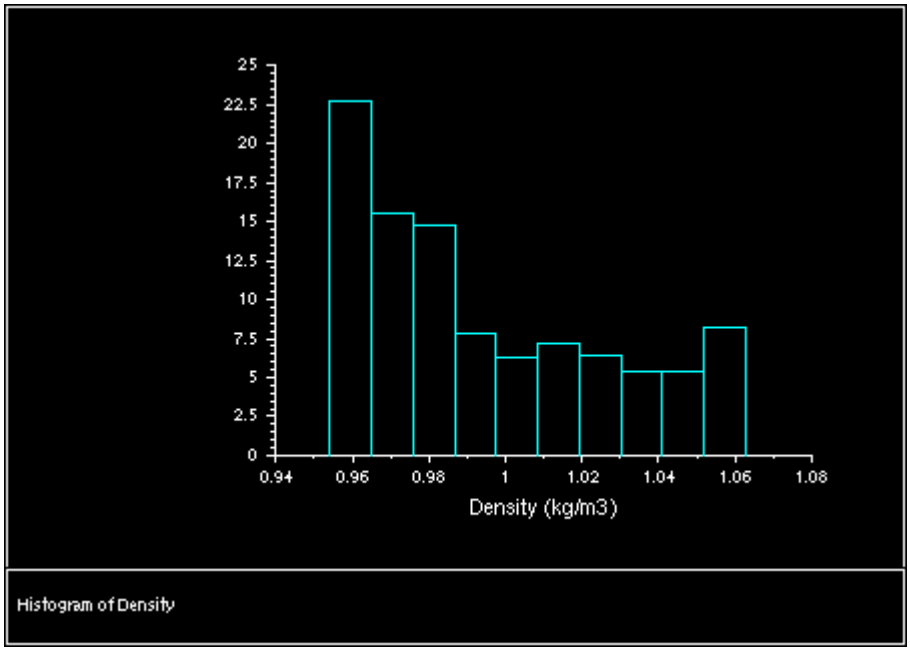
When an XY plot is displayed in the graphics window, you can use the **mouse-probe** option, but all other functions are inactive for XY plots. In addition, you can use any of the mouse buttons to move and resize the legend box.

31.9.1.2. Histograms

A histogram plot is a bar chart of data. It is a representation of a frequency of distribution by means of rectangles of widths representing class intervals and with areas proportional to the corresponding frequencies. When a histogram plot is displayed in the graphics window, you can use the **mouse-probe**, but all other functions are inactive for histogram plots. Figure 31.66: Sample Histogram (p. 1840) shows a sample histogram.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.66: Sample Histogram



See Histogram Reports (p. 1924) for information about printing histogram reports. For more information on histogram plots, see Histograms (p. 1851).

31.9.2. XY Plots of Solution Data

You can produce a very sophisticated XY plot by using data from several zones, surfaces, or files and modifying the axis and curve attributes. Using the capability for loading external data files, you can create plots that compare your ANSYS Fluent results with data from other sources. To get further information about the solution, you can investigate the frequency of distribution of the data using a histogram (see Histograms (p. 1851)).

31.9.2.1. Steps for Generating Solution XY Plots

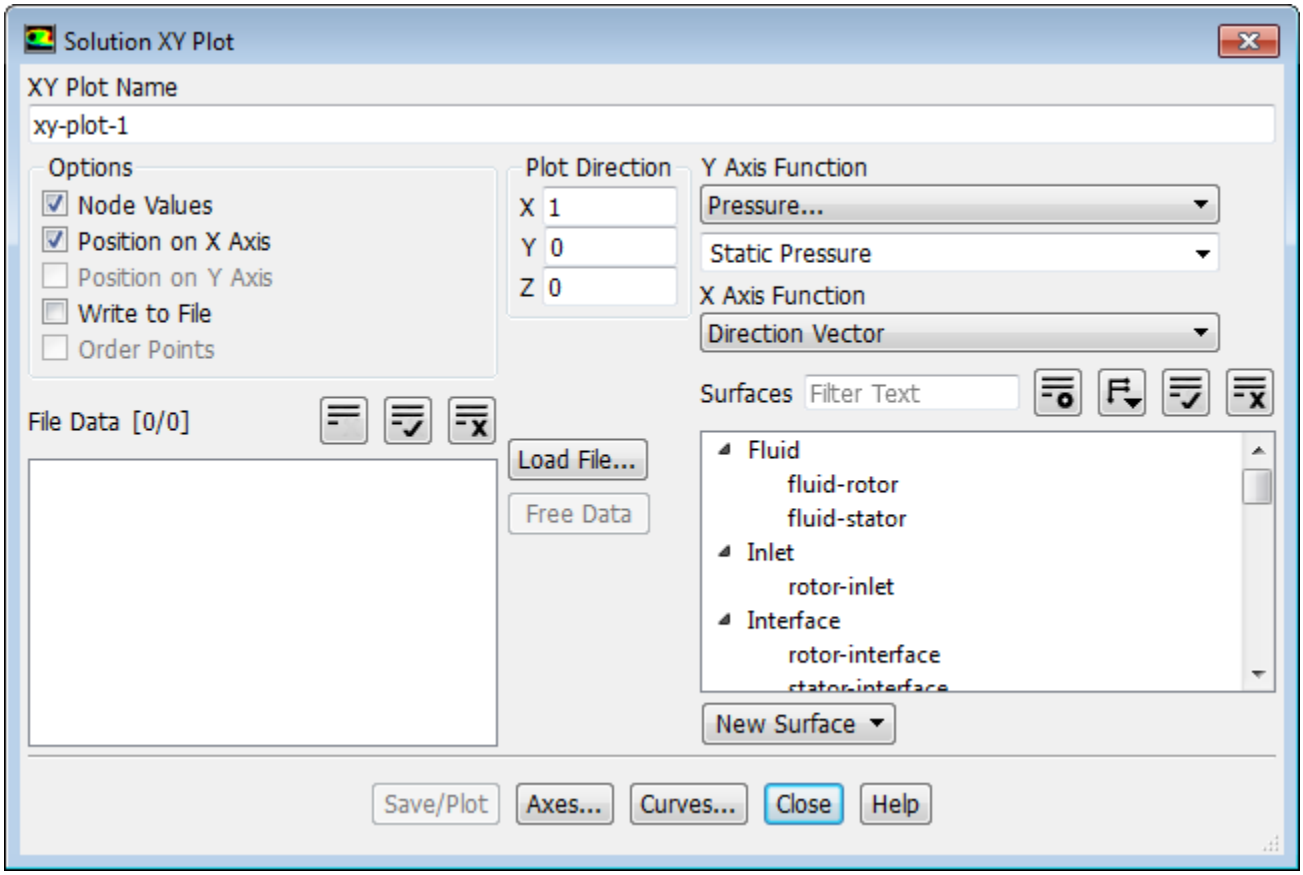
You can create an XY plot of solution data using the Solution XY Plot Dialog Box (p. 2595) (Figure 31.67: The Solution XY Plot Dialog Box (p. 1841)).

Named XY plot definitions can be saved for later use. See Creating and Using Contour Plot Definitions (p. 1762) for additional information on graphics object definitions.

Results → Plots → XY Plot New...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.67: The Solution XY Plot Dialog Box



The basic steps for generating a solution XY plot are as follows:

1. Specify the variable(s) you are plotting:

   - To plot a variable on the *y*-axis as a function of position on the *x*-axis, enable the **Position on X Axis** option and choose the variable to be plotted on the *y*-axis in the **Y Axis Function** drop-down list. Select a category from the upper list and then choose the desired quantity in the lower list. For an explanation of the variables in the list, see Field Function Definitions (p. 1931).

   - To plot a variable on the *x*-axis as a function of position on the *y*-axis, enable the **Position on Y Axis** option and choose the variable to be plotted on the *x*-axis in the **X Axis Function** drop-down list.

   - To plot one variable as a function of another, turn off both the **Position on X Axis** and **Position on Y Axis** options and select the variables to be plotted in the **X Axis Function** and **Y Axis Function** drop-down lists.

2. Specify the plot direction:

   - To plot a variable as a function of position along a specified direction vector, select **Direction Vector** in the **X Axis Function** or **Y Axis Function** drop-down list (whichever is the position axis), and specify the components of the direction vector for plotting under **Plot Direction**. The position axis of the plot is indicated by the selection of **Position on X Axis** or **Position on Y Axis**. The positions plotted will have coordinate values that correspond to the dot product of the data coordinate vector with the plot direction vector. For example, if you are plotting a variable at the pressure outlet of the geometry shown in Figure 31.68: Geometry Used for XY Plot (p. 1842), you would specify the **Plot Direction** vector (1,0,0)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

since you are interested in how the variable changes as a function of *x*. Figure 31.69: Data Plotted at Outlet Using a Plot Direction of (1,0,0) (p. 1843) shows the resulting XY plot. (If you specified (0,1,0) as the plot direction, all variable values would be plotted at the same position (see Figure 31.70: Data Plotted at Outlet Using a Plot Direction of (0,1,0) (p. 1843)), because the *y* value is the same at every point on the pressure outlet.)

- It is also possible to plot a variable as a function of position along the length of a specified curvilinear surface. The curvilinear surface must be piecewise linear and it cannot contain more than one closed curve, such as a complete circle. To plot a variable in this way, select **Curve Length** in the **X Axis Function** or **Y Axis Function** drop-down list (whichever is the position axis). Then specify the plot direction along the surface: to plot the variable along the direction of increasing curve length, select **Default** under **Plot Direction**; to plot the variable in the direction of decreasing surface length, select **Reverse**. To check the direction in which the variable will be plotted along a surface, select the surface in the **Surfaces** list and click **Show** under **Plot Direction**. ANSYS Fluent will display the selected surface in the graphics window, marking the start of the surface with a blue dot and the end of the surface with a red dot. ANSYS Fluent will also display arrows on the surface showing the direction in which the variable will be plotted.

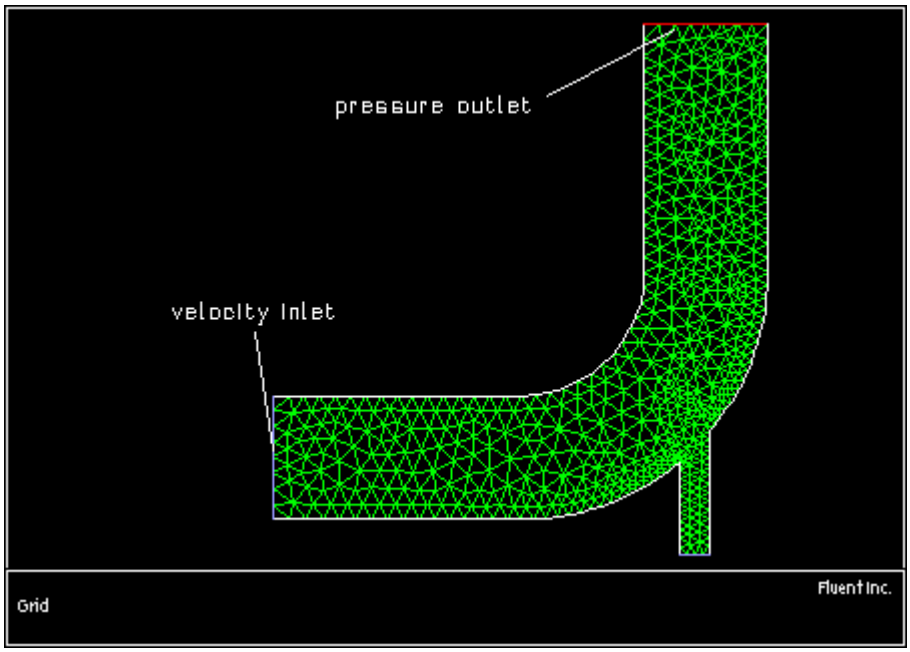
3. Choose the surface(s) on which to plot data in the **Surfaces** list. Note that if you are plotting a variable as a function of position along the length of a curvilinear surface, you can select only one surface in the **Surfaces** list.

4. Set any of the options described below, or modify the attributes of the axes or curves as described in Modifying Axis Attributes (p. 1852) and Modifying Curve Attributes (p. 1854).

5. Click **Plot** to generate the XY plot in the active graphics window.

You can use any of the mouse buttons to annotate the XY plot (see Adding Text to the Graphics Window (p. 1789)) or move the plot legend from its default position in the upper left corner of the graphics window.

Figure 31.68: Geometry Used for XY Plot



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.69: Data Plotted at Outlet Using a Plot Direction of (1,0,0)

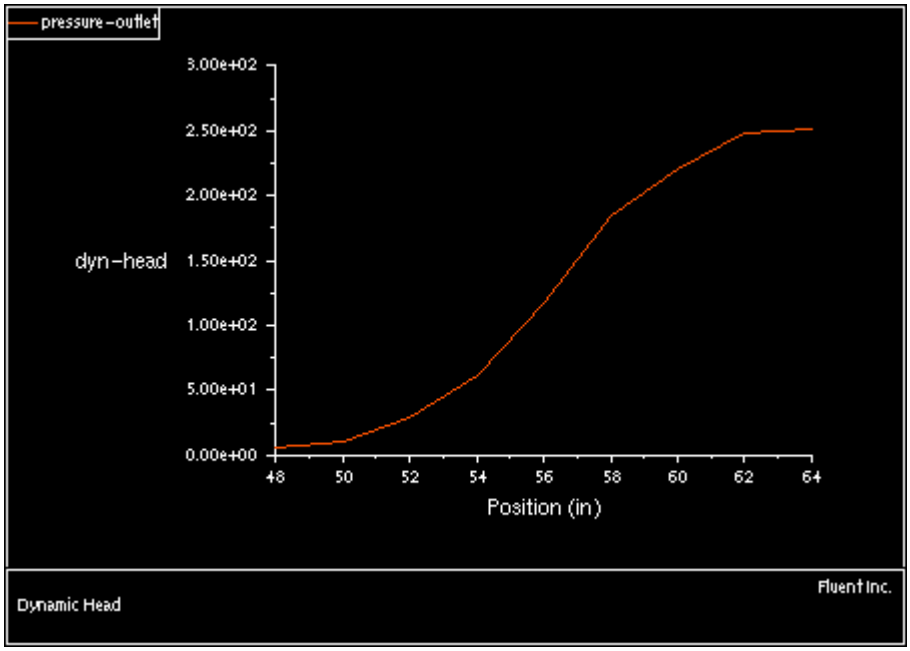
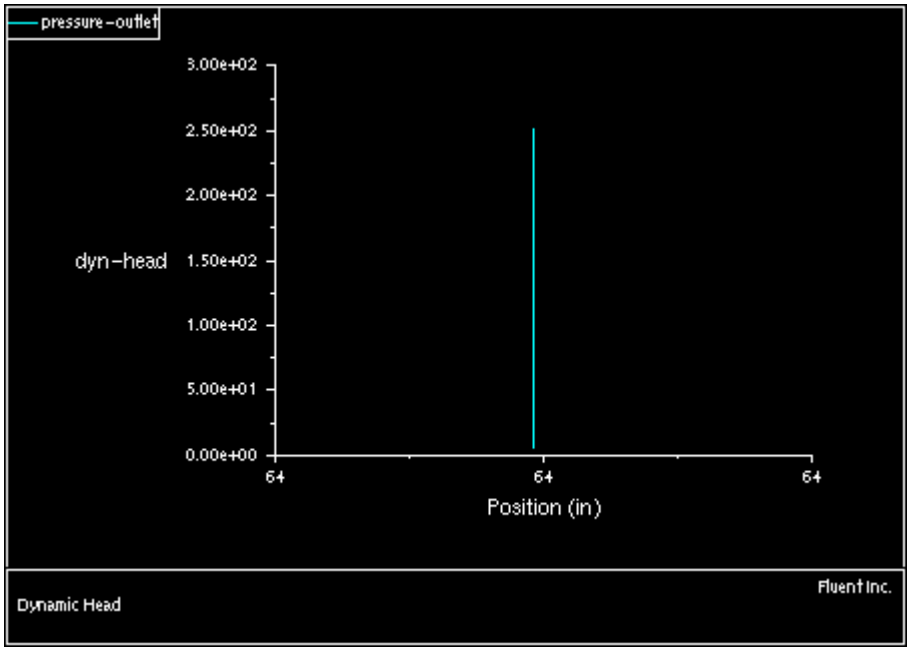


Figure 31.70: Data Plotted at Outlet Using a Plot Direction of (0,1,0)



31.9.2.2. Options for Solution XY Plots

The options mentioned in Steps for Generating Solution XY Plots (p. 1840) include the following.

- You can include data from an external file in the solution XY plot to compare your results with experimental data.
- You can also choose node or cell values to be plotted, and save the plot data to a file.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.9.2.2.1. Including External Data in the Solution XY Plot

To add external data to your XY plot for comparison with your results, you must first ensure that any external data files are in the format described in XY Plot File Format (p. 1850). You can then load the file(s) by clicking on the **Load File...** button and specifying the file(s) to be read in The Select File Dialog Box (p. 19). Once a file has been loaded, its title will appear in the **File Data** list. You can choose the data file(s) to be included in your plot from the titles in this list.

To remove a file from the **File Data** list, select it and then click **Free Data**.

31.9.2.2.2. Choosing Node or Cell Values

In ANSYS Fluent you can choose to display the computed cell-center values or values that have been interpolated to the nodes. By default, the **Node Values** option is turned on, and the interpolated values are displayed. If you prefer to display the cell values, turn the **Node Values** option off. Node-averaged data curves may be somewhat smoother than curves for cell values.

For face-only functions (for example, **Wall Shear Stress**), the cell values that are displayed for boundary zone surfaces will actually be the face values. These face values are more accurate, as face-only functions are computed on the faces and not on the cells. For these face-only functions, the cell values on post-processing surfaces will display the values in the cell. For more information about cell values, see Cell Values (p. 1931).

If you are displaying the XY plot to show the effect of a porous medium or 2D fan boundary, to depict a shock wave, or to show any other discontinuities or jumps in the plotted variable, you should use cell values; if you use node values in such cases, the discontinuity will be smeared by the node averaging for graphics and will not be shown clearly in the plot.

31.9.2.2.3. Saving the Plot Data to a File

Once you have generated an XY plot, you may want to save the plot data to a file. You can read this file into ANSYS Fluent at a later time and plot it alone using the File XY Plot Dialog Box (p. 2599), as described in XY Plots of File Data (p. 1844), or add it to a plot of solution data, as described above.

To save the plot data to a file, enable the **Write to File** option in the Solution XY Plot Dialog Box (p. 2595). The **Plot** button will change to the **Write...** button. Clicking on the **Write...** button will invoke The Select File Dialog Box (p. 19), in which you can specify a name and save a file containing the plot data. The format of this file is described in XY Plot File Format (p. 1850).

To sort the saved plot data in order of ascending *x*-axis value, enable the **Order Points** option in the Solution XY Plot Dialog Box (p. 2595) before you click **Write...**. This option is available only when the **Write to File** option is enabled.

31.9.3. XY Plots of File Data

You can produce XY plots using data contained in external files. The File XY Plot Dialog Box (p. 2599) enables you to display data read from external files in an abscissa/ordinate plot form. The format of the plot file is described in XY Plot File Format (p. 1850).

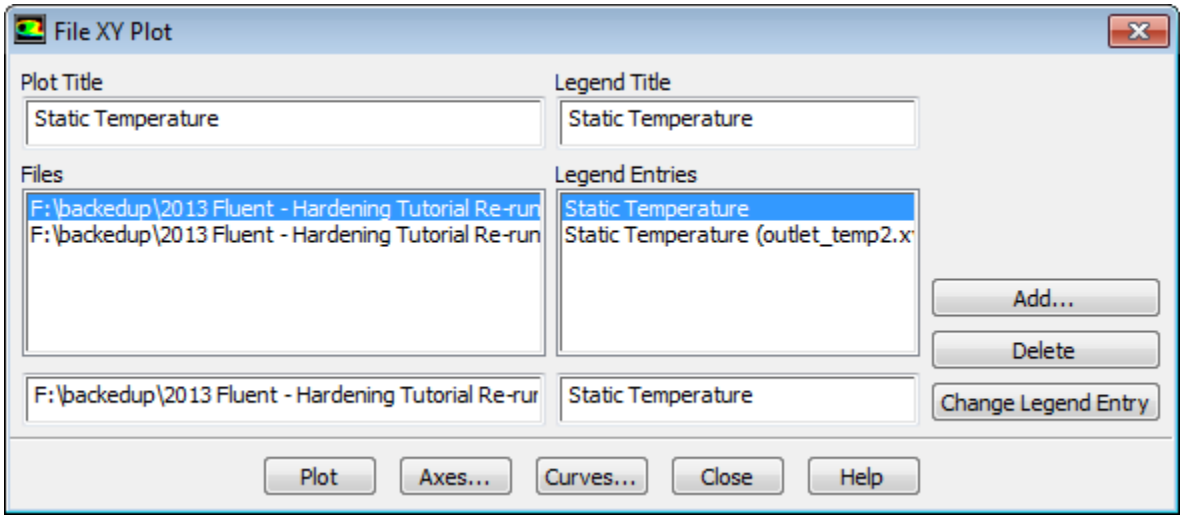
31.9.3.1. Steps for Generating XY Plots of Data in External Files

You can create an XY plot of data contained in one or more external files using the File XY Plot Dialog Box (p. 2599) (Figure 31.71: The File XY Plot Dialog Box (p. 1845)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Results → Plots → File → Set Up...

Figure 31.71: The File XY Plot Dialog Box



The steps for generating a file XY plot are as follows:

1. Load each external data file (with the format described in XY Plot File Format (p. 1850)) by entering its name in the text field beneath the **Files** list and clicking **Add...** (or pressing **Enter**). If you click **Add...** without specifying a name under **Files** (or if you specify an incorrect or duplicate name), The Select File Dialog Box (p. 19) will appear and you can specify one or more files there. When a file is loaded, its name will appear in the **Files** list and its title will appear in the **Legend Entries** list. Data in all loaded files will be plotted, so if you decide not to include one of the loaded files in the plot you must select it and click **Delete**.

2. Set any of the options described below, or modify the attributes of the axes or curves as described in Modifying Axis Attributes (p. 1852) and Modifying Curve Attributes (p. 1854).

3. Click **Plot** to generate an XY plot of the data associated with all loaded files.

31.9.3.2. Options for File XY Plots

The options mentioned in the procedure above include the following. You can change the plot title, legend title, or legend entry.

31.9.3.2.1. Changing the Plot Title

The plot title will appear in the caption box at the bottom of the graphics window. You can modify the plot title by changing the text in **Plot Title** in the File XY Plot Dialog Box (p. 2599) (or by editing the caption box manually, as described in Changing the Legend Display (p. 1788)).

31.9.3.2.2. Changing the Legend Entry

When you plot data from a single file, the y-axis of the plot will be labeled by the "legend entry." To modify this label, click the text in the **Legend Entries** list, edit the text that appears in the text field below the list, and then click **Change Legend Entry** (or press **Enter**). When you next plot the data, the new legend entry will appear in the plot.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.9.3.2.3. Changing the Legend Title

When you plot data from more than one file, a legend will appear in the upper left corner of the graphics window. By default, the legend will have no title. If you want to add a title, enter it in the **Legend Title** text field. The title will appear above the legend the next time you plot the data.

Note that you can use any of the mouse buttons to annotate the plot (see Adding Text to the Graphics Window (p. 1789)) or move the legend from its default position.

31.9.4. XY Plots of Profiles

ANSYS Fluent enables two options for generating XY plots of data related to boundary profiles. Using the Plot Profile Data Dialog Box (p. 2601), you can plot the original data points from the profile file you have read into ANSYS Fluent. Alternatively, you can plot the values assigned to the cell faces on the boundary after the profile file has been interpolated, by using the Plot Interpolated Data Dialog Box (p. 2601).

Important

Note that you must have valid data when trying to use the profile plotting options.

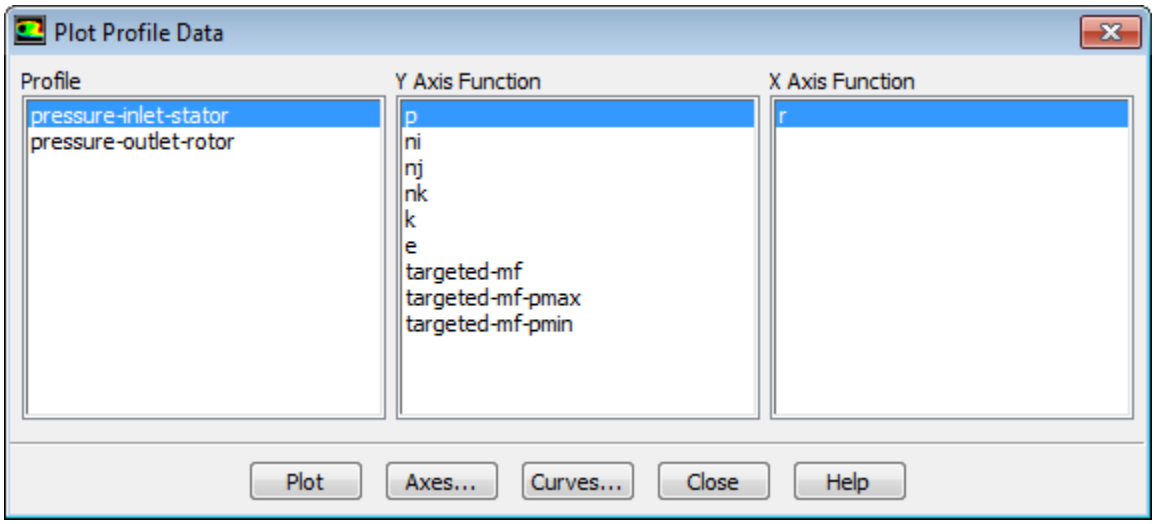
For more information about boundary profiles, see Profiles (p. 435).

31.9.4.1. Steps for Generating Plots of Profile Data

Once you have read a profile file, it is available for plotting by using the Plot Profile Data Dialog Box (p. 2601).

Results → Plots → Profile Data → Set Up...

Figure 31.72: The Plot Profile Data Dialog Box



The procedure for generating an XY plot of the original profile data is as follows:

1. Select one of the profiles you have read from the **Profile** selection list.

2. Select a field of the profile from the **Y Axis Function** selection list.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Choose a variable against which you want to plot the field data, and select it from the **X Axis Function** selection list. The available variables will vary depending on the profile, and include **x**, **y**, **z**, **r**, and **time**.

4. Modify the attributes of the axes or curves as described in Modifying Axis Attributes (p. 1852) and Modifying Curve Attributes (p. 1854).

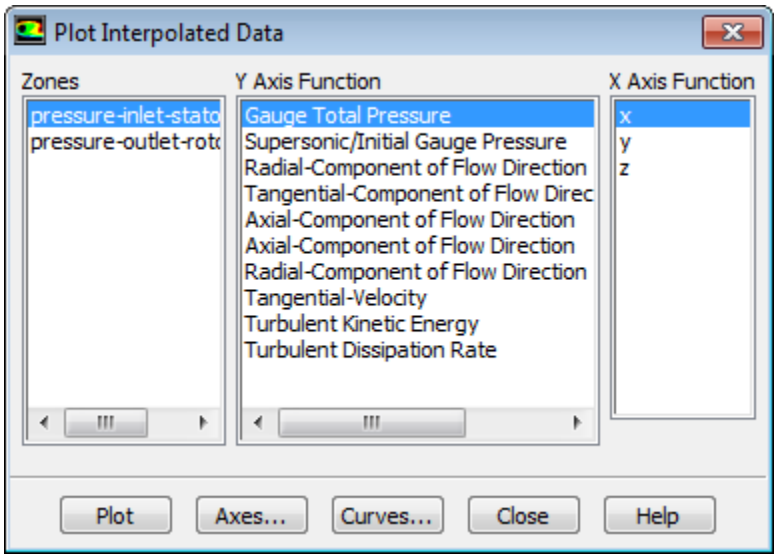
5. Click **Plot** to generate an XY plot of the profile field data.

31.9.4.2. Steps for Generating Plots of Interpolated Profile Data

To interpolate a profile you must first read a profile file for the case, and select a profile field in a boundary conditions dialog box (for example, the Velocity Inlet Dialog Box (p. 2447)). After the flow solution has been initialized, the cell face values of the boundary zone can be plotted by using the Plot Interpolated Data Dialog Box (p. 2601).

Results → Plots → Interpolated Data → Set Up...

Figure 31.73: The Plot Interpolated Data Dialog Box



The procedure for generating an XY plot of the interpolated data is as follows:

1. Select a zone from the **Zones** selection list. Only the zones for which you have set a profile field as one or more of the parameters will be available in this list.

2. Select a profile-related parameter of the zone from the **Y Axis Function** selection list. The name of the parameter will be the same as that of the drop-down list in the boundary condition dialog box from which the profile field was selected.

3. Choose a variable against which you want to plot the field data, and select it from the **X Axis Function** selection list. The available variables are **x**, **y**, and (for 3D cases) **z**.

4. Modify the attributes of the axes or curves as described in Modifying Axis Attributes (p. 1852) and Modifying Curve Attributes (p. 1854).

5. Click **Plot** to generate an XY plot of the cell face values on the boundary.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.9.5. XY Plots of Circumferential Averages

You can also generate a plot of circumferential averages in ANSYS Fluent. This enables you to find the average value of a quantity at several different radial or axial positions in your model. ANSYS Fluent computes the average of the quantity over a specified circumferential area, and then plots the average against the radial or axial coordinate.

31.9.5.1. Steps for Generating an XY Plot of Circumferential Averages

You can generate an XY plot of circumferential averages in the radial direction using the `circum-avg-radial` text command:

`plot` → `circum-avg-radial`

or you can use the `circum-avg-axial` text command to generate an average in the axial direction:

`plot` → `circum-avg-axial`

The procedure for generating an XY plot of circumferential averages is as follows:

1. Specify the variable to be averaged by typing its name when ANSYS Fluent prompts you for `averages of`. You can press **Enter** to see a list of available variables.

2. Choose the surface on which to plot data by typing its name when ANSYS Fluent prompts you for `on surface`.

   Important

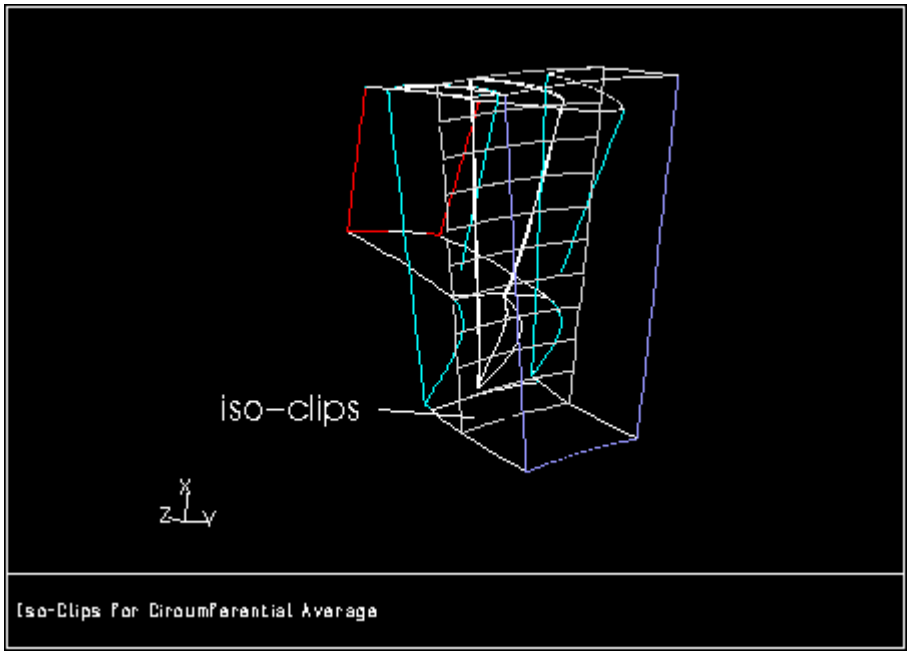
   Use the Mesh Display Dialog Box (p. 2163) to see a list of surfaces on which you can plot data. Pressing **Enter** will not show a list of available surfaces.

3. Specify the `number of bands` to be created. The default number of bands is 5.

   ANSYS Fluent will create circumferential bands by isoclipping the specified surface into equal bands of radial or axial coordinate. An example of the iso-clips created is shown in Figure 31.74: Iso-Clips Created For Circumferential Averaging (p. 1849). The radial or axial coordinate is derived from the rotation axis of the **Reference Zone** specified in the Reference Values Task Page (p. 2502).

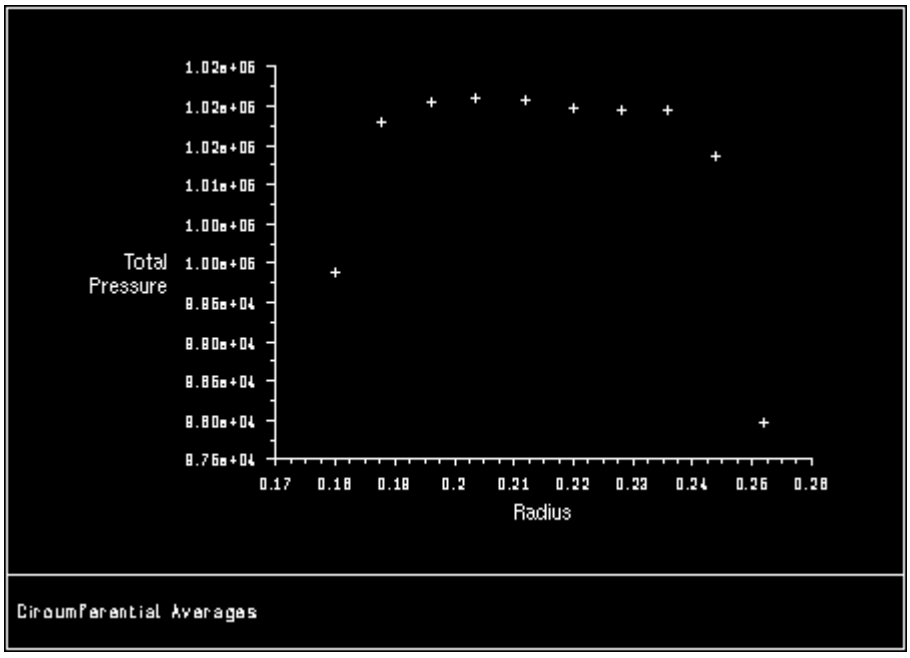
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.74: Iso-Clips Created For Circumferential Averaging



ANSYS Fluent then computes the average of the variable for each band using the area-weighted average described in Computing Surface Integrals of the Theory Guide. Finally, it plots the average of the variable as a function of radial or axial coordinate. Figure 31.75: XY Plot of Circumferential Averages (p. 1849) shows an example of an XY plot of circumferential averages using radial coordinates.

Figure 31.75: XY Plot of Circumferential Averages



When the circumferential average plot is generated, ANSYS Fluent also creates a new surface called `radial-bands` or `axial-bands`, which contains the iso-clips described above (see Figure 31.74: Iso-Clips Created For Circumferential Averaging (p. 1849)). You can use this surface to generate other XY plots. For more information on the creation and manipulation of surfaces, see Creating Surfaces and Cell Registers for Displaying and Reporting Data (p. 1709).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.9.5.2. Customizing the Appearance of the Plot

If you want to customize the appearance of the axes or curves in a circumferential average plot, you can save the plot data to a file (using the `plot-to-file` text command, as described in XY Plot File Format (p. 1850)), read the file into ANSYS Fluent and plot it again (using the File XY Plot Dialog Box (p. 2599), as described in XY Plots of File Data (p. 1844)), and then use the Axes Dialog Box (p. 2608) and Curves Dialog Box (p. 2610) (as described in Modifying Axis Attributes (p. 1852) and Modifying Curve Attributes (p. 1854)) to modify the appearance of the plot.

To save the plot data to a file, first use the `plot-to-file` text command to specify the name of the file.

`plot` → `file-set` → `plot-to-file`

Then generate the circumferential average XY plot as described above. ANSYS Fluent will display the plot in the graphics window, and also save the plot data to the specified file.

31.9.6. XY Plot File Format

The XY file format read or written by ANSYS Fluent includes the following information:

- The title of the plot
- The label for the abscissa and the ordinate
- Cortex variables and pairs of abscissa/ordinate data for each curve in the plot

The following sample file illustrates the XY file format:

```
(title "Velocity Magnitude")
(labels "Position" "Velocity Magnitude")

((xy/key/label "pressure-inlet-8")
  (xy/key/visible? #t)
  (xy/line/pattern "--")
  0.0000 230.097 0.
  0625 160.551
  0.1250 149.205
  ...
  0.5000 183.007
)
```

Similar to the case file format, parentheses bound the various pieces of information in the formatted, ASCII file. The title (`title " "`) and labels (`labels " "`) must be first in the file, then each curve has information in the form (`(cxvar value) x y x y x y...`), where there may be zero or more Cortex variables defined for each curve.

You do not have to include Cortex variables to import your XY data. For example, you may want to import experimental data to compare with the ANSYS Fluent solution. The following example would use the default Cortex variables in the code to define the data. After you import the file into ANSYS Fluent, you could then use the Axes Dialog Box (p. 2608) and the Curves Dialog Box (p. 2610) to customize the XY plot, as described in Modifying Axis Attributes (p. 1852) and Modifying Curve Attributes (p. 1854).

```
(title "Experiment, Run 11")
(labels "X, m" "Cp")
( 0 1.5
  1.5 1.3
  3.2 1.5
  5.1 1.2
)
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.9.7. Residual Plots

Residual history can be displayed using an XY plot. The abscissa of the plot corresponds to the number of iterations and the ordinate corresponds to the log-scaled residual values.

To plot the current residual history, click **Plot** in the Residual Monitors Dialog Box (p. 2789).

Solution → Monitors → Residuals Edit...

For additional information about using the Residual Monitors Dialog Box (p. 2789) to plot residuals, see Printing and Plotting Residuals (p. 1627).

31.9.8. Histograms

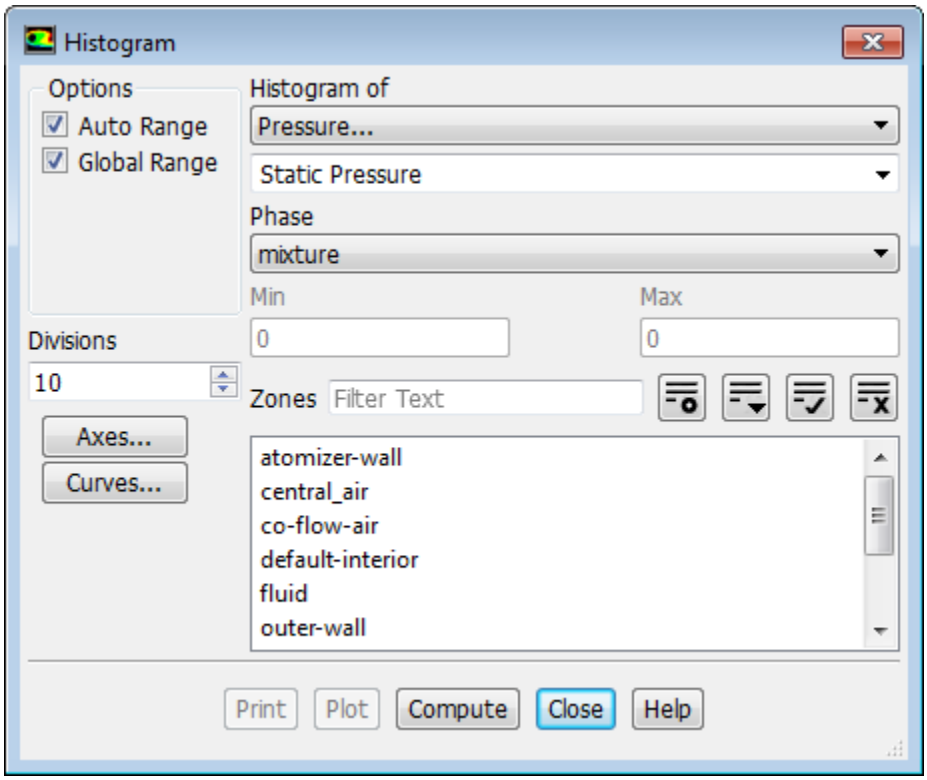
Histograms can be displayed in a graphics window using a bar chart (or printed in the console window, as described in Histogram Reports (p. 1924)). The abscissa of the chart is the desired solution quantity and the ordinate is the percentage of the total number of cells.

31.9.8.1. Steps for Generating Histogram Plots

You can create a histogram plot of solution data using the Histogram Dialog Box (p. 2598) (Figure 31.76: The Histogram Dialog Box (p. 1851)).

Results → Plots → Histogram Edit...

Figure 31.76: The Histogram Dialog Box



The steps for generating a histogram plot are as follows:

1. Choose the scalar quantity to be plotted in the **Histogram Of** drop-down list. Select a category in the upper list and then select the desired quantity in the lower list. (See Field Function Definitions (p. 1931) for an explanation of the variables in the list.)

2. Set the number of data intervals that will be plotted in the histogram in the **Divisions** field. By default there will be 10 intervals ("bars") in the histogram plot to finer intervals, increase the number of **Divisions**. You may want to click **Compute** to update the **Min** and **Max** fields when you are trying to decide how many divisions to plot.

3. Select the face or cell zone under **Zones** for which you want results plotted or printed. If all zones are selected, then the entire domain will be plotted. You can also plot histograms based on the selected **Zone Types**.

4. Set the option described below, if desired, or modify the attributes of the axes or curves as described in Modifying Axis Attributes (p. 1852) and Modifying Curve Attributes (p. 1854).

5. Click **Plot** to generate the histogram plot in the active graphics window.

6. Click **Print** to print out your histogram results on individual zones, or the entire domain. Similarly, you can click **Compute** to calculate your histogram results on individual zones, or the entire domain.

31.9.8.2. Options for Histogram Plots

Other than the axis and curve attribute controls mentioned in the procedure above, the only option for histogram plotting is the ability to specify a subrange of values to be plotted.

31.9.8.2.1. Specifying the Range of Values Plotted

By default, the range of values included in the histogram plot is automatically set to the range of values in the entire domain for the selected variable. If you want to focus in on a smaller range of values, you can restrict the range to be displayed.

To manually set the range of values, turn off the **Auto Range** option in the Histogram Dialog Box (p. 2598). The **Min** and **Max** fields will become editable, and you can enter the new range of values to be plotted. To show the default range at any time, click **Compute** and the **Min** and **Max** fields will be updated.

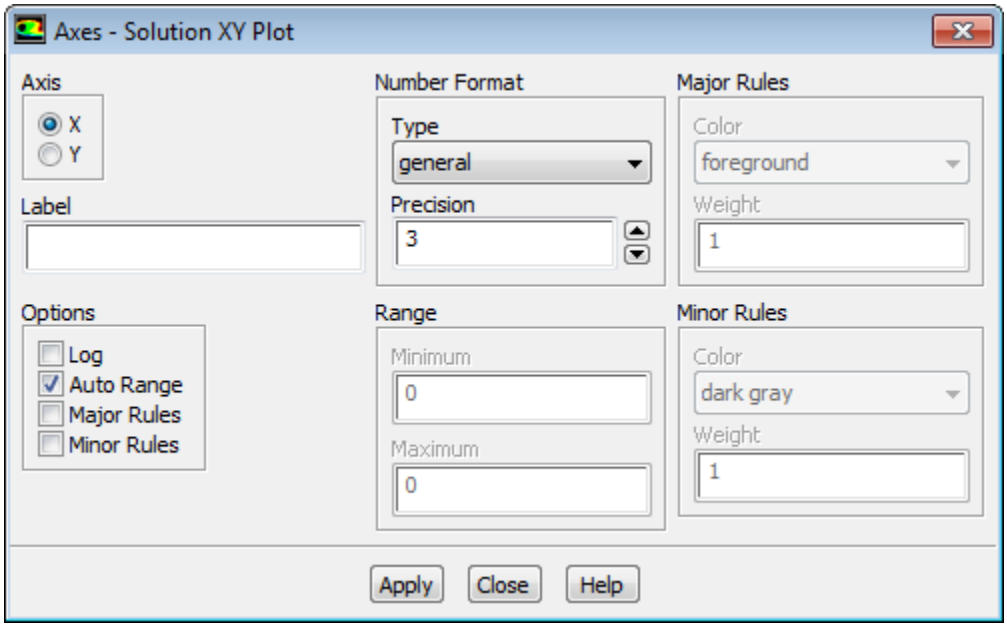
You can also choose to base the minimum and maximum values on the range of values on the selected surfaces, rather than in the entire domain. To do this, turn off the **Global Range** option in the Histogram Dialog Box (p. 2598). The **Min** and **Max** values will be updated when you next click **Compute**.

31.9.9. Modifying Axis Attributes

You can modify the appearance of the XY and parameters that control the labels, scale, range, numbers, and major and minor rules. For each type of plot (solution XY, file XY, profile, residual, histogram, and so on), you can set different axis parameters in the Axes Dialog Box (p. 2608) (Figure 31.77: The Axes Dialog Box (p. 1853)). Note that the title following **Axes** in the dialog box indicates which plot environment you are changing (for example, the **Axes - Solution XY Plot** dialog box controls parameters for solution XY plots).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.77: The Axes Dialog Box



To open the Axes Dialog Box (p. 2608) for a particular plot type, click **Axes...** in the appropriate dialog box (for example, the **Solution XY Plot**, **File XY Plot**, **Plot Profile Data**, **Plot Interpolated Data**, or **Residual Monitors** dialog box).

31.9.9.1. Using the Axes Dialog Box

The Axes Dialog Box (p. 2608) enables you to independently control the characteristics of the ordinate (*y*-axis) and abscissa (*x*-axis) on an XY plot or parameters for one axis or the other, by following the procedure below:

1. Choose the axis for which you want to modify the attributes by selecting **X** or **Y** under **Axis**.
2. Set the desired parameters.
3. Click **Apply** and then choose the other axis and repeat the steps, if desired.

The changes to the axis attributes will appear in the graphics window the next time you generate a plot.

31.9.9.1.1. Changing the Axis Label

If you want to modify the label for the axis, you can do so by editing the **Label** text field in the Axes Dialog Box (p. 2608).

31.9.9.1.2. Changing the Format of the Data Labels

You can change the format of the labels that define the primary data divisions on the axes using the controls under the **Number Format** heading in the Axes Dialog Box (p. 2608).

- To display the real value with an integral and fractional part (for example, 1.0000), select **float** in the **Type** drop-down list. You can set the number of digits in the fractional part by changing the value of **Precision**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To display the real value with a mantissa and exponent (for example, 1.0e-02), select **exponential** in the **Type** drop-down list. You can define the number of digits in the fractional part of the mantissa in the **Precision** field.

- To display the real value with either float or exponential form, depending on the size of the number and the defined **Precision**, choose **general** in the **Type** drop-down list.

31.9.9.1.3. Choosing Logarithmic or Decimal Scaling

By default, decimal scaling is used for both axes (except for the y-axis in residual plots, which uses a log scale). If you want to change to a logarithmic scale, enable the **Log** option in the Axes Dialog Box (p. 2608). To return to a decimal scale, turn off the **Log** option. Note that when you are using the logarithmic scale, the **Range** values are the exponents; to specify a logarithmic range from 1 to 10000, for example, you will specify a minimum value of 1 and a maximum value of 4.

31.9.9.1.4. Resetting the Range of the Axis

By default, the extents of the axis will range from the minimum value plotted to the maximum value plotted. If you want to change the range or extents of the axis, you can do so by turning off the **Auto Range** option in the Axes Dialog Box (p. 2608) and setting the new **Minimum** and **Maximum** values for the **Range**. This feature is useful when you are generating a series of plots and you want the extents of one or both of the axes to be the same, even if the range of plotted values differs. For example, if you are generating plots of temperature on several different wall zones, you might want the minimum and maximum temperature on the y-axis to be the same in every plot so that you can easily compare one plot with another. You would determine a temperature range that includes the temperatures on all walls, and use that as the range for the y-axis in each plot.

31.9.9.1.5. Controlling the Major and Minor Rules

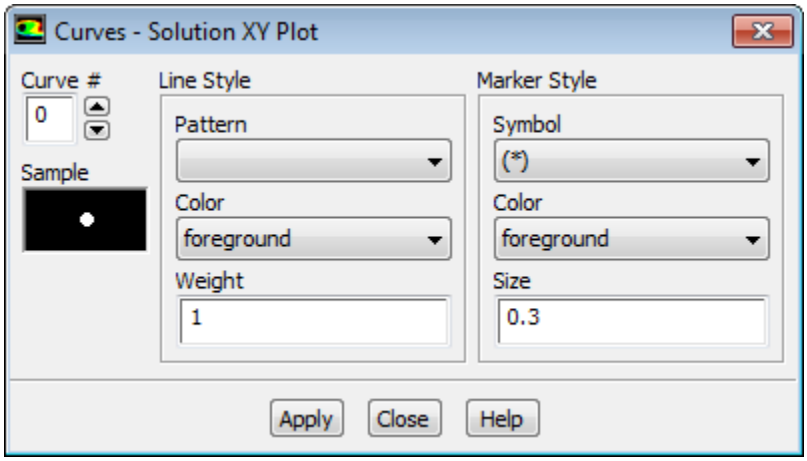
ANSYS Fluent enables you to display major and/or minor rules on the axes. Major and minor rules are the horizontal or vertical lines that mark, respectively, the primary and secondary data divisions and span the whole plot window to produce a "mesh." To add major or minor rules to the plot, enable the **Major Rules** or **Minor Rules** option. You can then specify a color and weight for each type of rule. Under the **Major Rules** or **Minor Rules** heading, select the desired color for the lines in the **Color** drop-down list and specify the line thickness in the **Weight** field. A line of weight 1.0 is normally 1 pixel wide. A weight of 2.0 would make the line twice as thick (that is, 2 pixels wide).

31.9.10. Modifying Curve Attributes

The data curves in XY plots and histograms can be represented by any combination of lines and markers. You can modify the attributes of the curves, including the patterns, weights, and colors of the lines, and the symbols, sizes, and colors of the markers. For each type of plot (solution XY, file XY, profile, residual, parameters in the Curves Dialog Box (p. 2610) (Figure 31.78: The Curves Dialog Box (p. 1855)). Note that the title following **Curves** in the dialog box indicates which plot environment you are changing (for example, the **Curves - Solution XY Plot** dialog box controls curve parameters for solution XY plots).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.78: The Curves Dialog Box



To open the Curves Dialog Box (p. 2610) for a particular plot type, click **Curves...** in the appropriate dialog box (for example, **Solution XY Plot**, **File XY Plot**, **Plot Profile Data**, **Plot Interpolated Data**, or **Residual Monitors** dialog box).

31.9.10.1. Using the Curves Dialog Box

The Curves Dialog Box (p. 2610) enables you to independently control the characteristics of each data curve in an XY plot or parameters for a curve, you will follow the procedure below:

1. Specify the curve for which you want to modify the attributes by increasing or decreasing the **Curve #** counter. The curves are numbered sequentially, starting from 0. For example, if you were plotting flow-field values on two surfaces, the first surface would be curve 0, and the second, curve 1. If the plot contains only one curve, the **Curve #** is set to 0 and is not editable.

2. Set the desired line and/or marker parameters as described below.

3. Click **Apply** and then choose another **Curve #** and repeat the steps, if desired.

Your changes to the curve attributes will appear in the graphics window the next time you generate a plot.

31.9.10.1.1. Changing the Line Style

You can control the pattern, color, and weight of the line using the controls under the **Line Style** heading:

- To set the line pattern for the curve, choose one of the items in the **Pattern** drop-down list. Except for **center** and **phantom** lines, the list displays examples of the pattern choices. A **center** line alternates a very long dash and a short dash and a **phantom** line alternates a very long dash and a double short dash. Note that selecting the second item in the drop-down list, represented by 4 short dashes, will result in a solid-line curve.

> **Important**
>
> If you do not want the data points to be connected by any type of line (that is, if you plan to use just markers), select the "blank" choice, which is the first item in the **Pattern** list.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To set the color of the line, pick one of the choices in the **Color** drop-down list.
- To define the line thickness, set the value of **Weight**. A line weight of 1.0 is normally 1 pixel wide. Therefore, a weight of 2.0 would make the line twice as thick (that is, 2 pixels wide).

31.9.10.1.2. Changing the Marker Style

You can control the symbol, color, and size for the data marker using the controls under the **Marker Style** heading:

- To set the symbol used to mark data, choose one of the items in the **Symbol** drop-down list. The list displays examples of the symbol choices. For example, in plotting pressure-coefficient data on the upper and lower surfaces of an airfoil, the symbol /*\ (filled-in upward-pointing triangle) could be used for the marker representing the upper surface data, and the symbol */ (filled-in downward-pointing triangle) could be used for the marker representing the lower surface data.

  Important

  If you do not want the data points to be represented by markers (that is, if you plan to use just a line connecting the data points), select the "blank" choice, which is the first item in the **Style** list.

- To set the color of the marker, pick one of the choices in the **Color** drop-down list.
- To define the size of the data marker, set the value of **Size**. A symbol of size 1.0 is 3.0% of the height of the display screen, except for the "." symbol, which is always one pixel.

31.9.10.1.3. Previewing the Curve Style

To see what a particular setting will look like in the plot, you can preview it in the **Sample** window of the Curves Dialog Box (p. 2610). A single marker and/or line will be shown with the specified style attributes.

31.10. Turbomachinery Postprocessing

In addition to the many graphics tools already discussed, ANSYS Fluent also provides turbomachinery-specific postprocessing features that can be accessed once you have defined the topology of the problem. Information on postprocessing for turbomachinery applications is provided in the following sections:

31.10.1. Defining the Turbomachinery Topology
31.10.2. Generating Reports of Turbomachinery Data
31.10.3. Displaying Turbomachinery Averaged Contours
31.10.4. Displaying Turbomachinery 2D Contours
31.10.5. Generating Averaged XY Plots of Turbomachinery Solution Data
31.10.6. Globally Setting the Turbomachinery Topology
31.10.7. Turbomachinery-Specific Variables

31.10.1. Defining the Turbomachinery Topology

In order to establish the turbomachinery-specific coordinate system used in subsequent postprocessing functions, ANSYS Fluent requires you to define the topology of the flow domain. The procedure for defining the topology is described further in this section, along with details about the boundary types.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The current implementation of the turbomachinery topology definition for postprocessing is no longer limited to one row of blades at a time. If your geometry contains multiple rows of blades, you can define all turbomachinery topologies simultaneously. You can name and/or manage all topologies and perform various turbomachinery postprocessing tasks on a single topology or on all topologies at once.

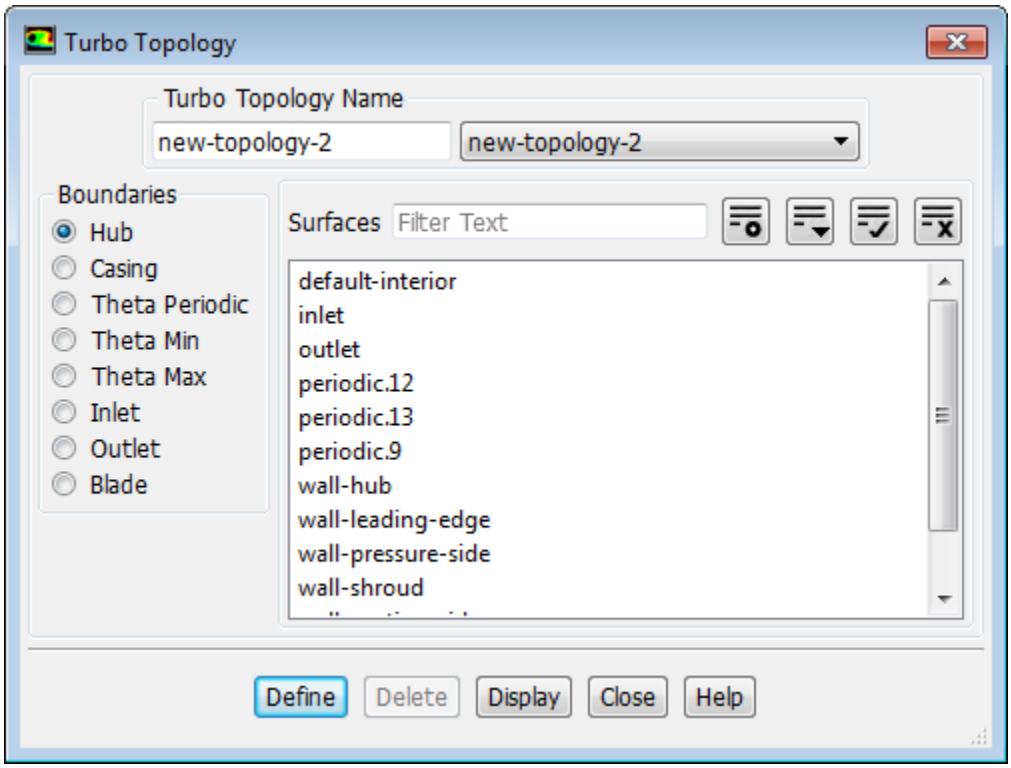
Important

The turbo coordinates can only be generated properly if the correct rotation axis is specified in the boundary conditions dialog box for the fluid zone (see Specifying the Rotation Axis (p. 255)).

To define the turbomachinery topology in ANSYS Fluent, you will use the Turbo Topology Dialog Box (p. 2822) (Figure 31.79: The Turbo Topology Dialog Box (p. 1857)).

Setting Up Domain → Mesh Models → Turbo Topology...

Figure 31.79: The Turbo Topology Dialog Box



The procedure for defining topology for your turbomachinery application are as follows:

1. Select a boundary type under **Boundaries** (for example, **Hub** in Figure 31.79: The Turbo Topology Dialog Box (p. 1857)). The boundary types are described in detail in Boundary Types (p. 1858).

2. In the **Surfaces** list, choose the surface(s) that represent the boundary type you selected in step 1.

   If you want to select several surfaces of the same type, click and select **Surface Type** under **Group By**, which organizes the surfaces in a tree view that is grouped by surface type.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Another shortcut is to use the **Filter Text** entry box to filter the **Surfaces** list to show only the surfaces that match the pattern you enter. For additional information on using the **Filter Text** entry box, see Filter Text Entry Boxes (p. 15).

3. Repeat the steps 1 and 2 for all the boundary types that are relevant for your model.

Important

For a complete turbo topology definition the surfaces defined as inlet, outlet, hub, casing, periodic, theta min, and theta max (if available) should form a closed domain.

4. Enter a name in the **Turbo Topology Name** field or keep the default name.

5. Click **Define** to complete the definition of the boundaries.

   ANSYS Fluent will inform you that the turbomachinery postprocessing functions have been activated.

6. Specify a position vector that is defined as $\theta=0$. This position vector should be outside the domain, for example, if your domain lies in the first and second quadrant, specify negative y-axis as the zero θ line. This will ensure that there is no discontinuity in angular coordinates within the domain. This can be done using the `display/set/zero-angle-dir` command.

   Default zero $\theta=0$ line is $+y$-axis. If this axis passes through the domain, you should define the zero θ line, so as to satisfy above criteria.

7. To view a defined topology, select the topology from the **Turbo Topology Name** drop-down list and click **Display**. The defined topology is shown in the active graphics window. This enables you to visually check the boundaries to ensure that you have defined them correctly.

8. To edit a defined topology, select the topology from the **Turbo Topology Name** drop-down list, make the appropriate changes and click **Modify**.

9. To remove a defined topology, select the topology from the **Turbo Topology Name** drop-down list and click **Delete**.

Important

Note that the topology setup that you define will be saved to the case file when you save the current model. Thus, if you read this case back into ANSYS Fluent, you do not need to set up the topology again.

However, use of a boundary condition file to set the turbo topology for two similar cases may not work properly. In that case you need to set the turbo topology manually.

31.10.1.1. Boundary Types

The boundaries for the turbomachinery topology are as follows (see Figure 31.80: Turbomachinery Boundary Types (p. 1859)):

Hub
is the wall zone(s) forming the lower boundary of the flow passage (generally toward the axis of rotation of the machine).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Casing
is the wall zone(s) forming the upper boundary of the flow passage (away from the axis of rotation of the machine).

Theta Periodic
is the periodic boundary zone(s) on the circumferential boundaries of the flow passage.

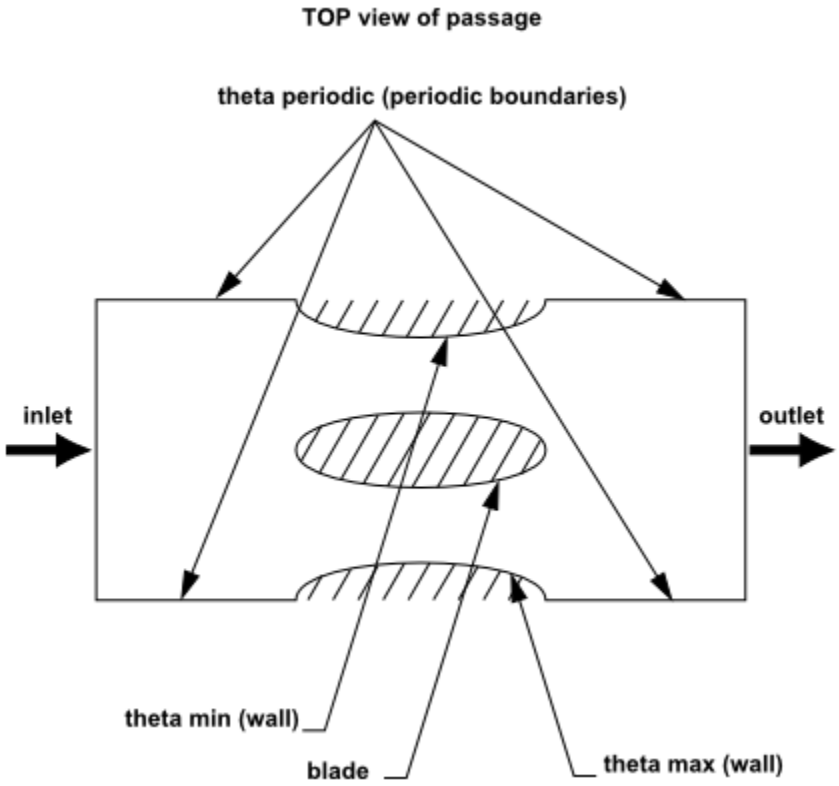
Theta Min
and **Theta Max** are the wall zones at the minimum and maximum angular (θ) positions on a circumferential boundary.

Inlet
is the inlet zone(s) through which the flow enters the passage.

Outlet
is the outlet zone(s) through which the flow exits the passage.

Blade
is the wall zone(s) that defines the blade(s) (if any). Note that these zones cannot be attached to the circumferential boundaries. For this situation, use **Theta Min** and **Theta Max** to define the blade.

Figure 31.80: Turbomachinery Boundary Types



31.10.2. Generating Reports of Turbomachinery Data

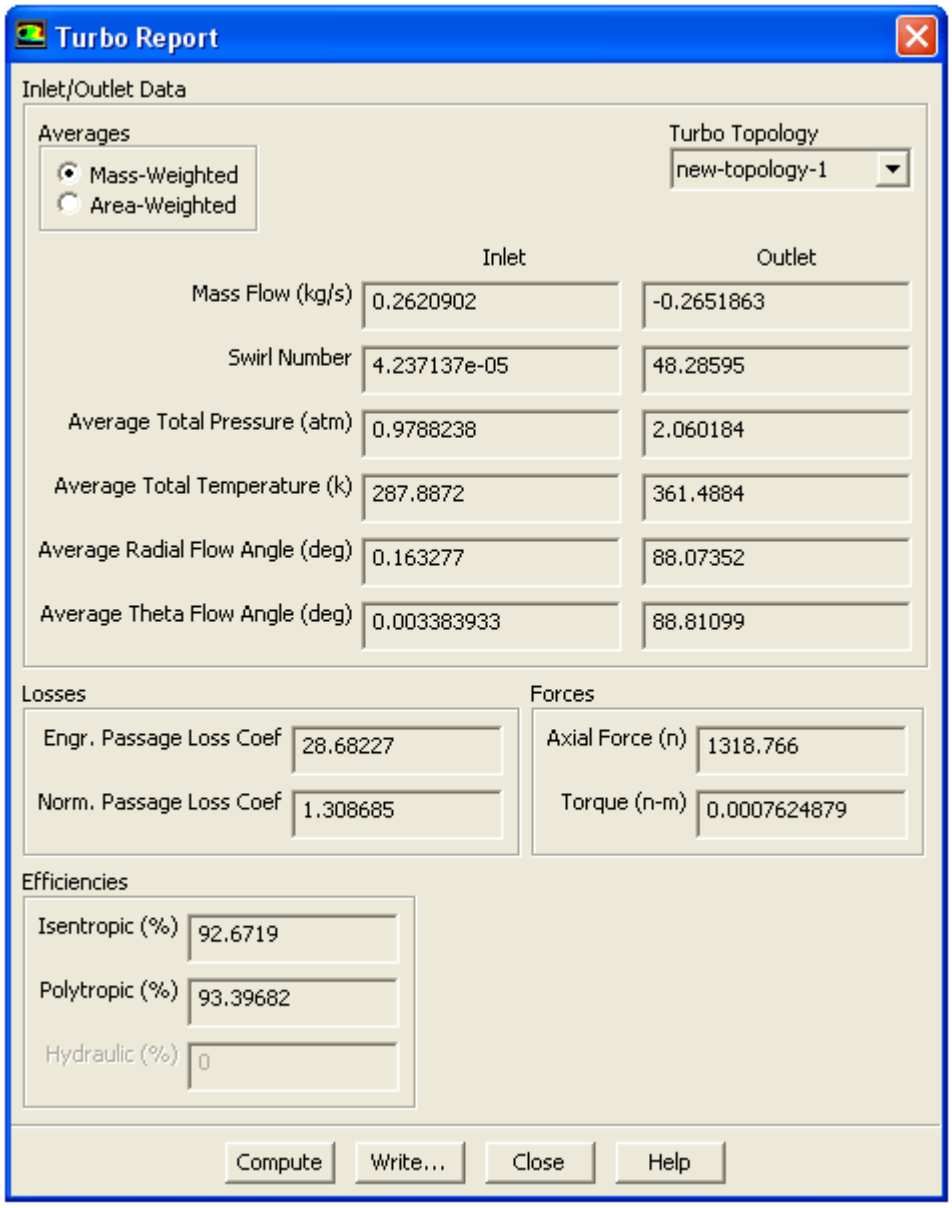
Once you have defined your turbomachinery topologies, as described in Defining the Turbomachinery Topology (p. 1856), you can report a number of turbomachinery quantities, including mass flow, swirl number, torque, and efficiencies.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To report turbomachinery quantities in ANSYS Fluent, you will use the Turbo Report Dialog Box (p. 2819) (Figure 31.81: The Turbo Report Dialog Box (p. 1860)).

Postprocessing → Model Specific → Turbo Topology → Report...

Figure 31.81: The Turbo Report Dialog Box



The procedure for using this dialog box is as follows:

1. Under **Averages**, specify whether you want to report **Mass-Weighted** or **Area-Weighted** averages.
2. Under **Turbo Topology**, specify a predefined turbomachinery topology from the drop-down list.
3. Click **Compute**. ANSYS Fluent will compute the turbomachinery quantities as described below, and display their values.
4. If you want to save the reported values to a file, click **Write...** and specify a name for the file in The Select File Dialog Box (p. 19).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.10.2.1. Computing Turbomachinery Quantities

31.10.2.1.1. Mass Flow

The mass flow rate through a surface is defined as follows:

$$\dot{m}=\int_A (\rho\vec{v}\cdot\hat{n})dA \tag{31.2}$$

where

A = is the area of the inlet or outlet

$\vec{v}$ = the velocity vector

ρ = the fluid density

$\hat{n}$ = a unit vector normal to the surface

31.10.2.1.2. Swirl Number

The swirl number is defined as follows:

$$SW=\frac{\int_S rv_\theta(\vec{v}\cdot\hat{n})\,dS}{\overline{r}\int_S v_z(\vec{v}\cdot\hat{n})\,dS} \tag{31.3}$$

where

r = the radial coordinate (specifically, the radial distance from the axis of rotation)

v_θ = the tangential velocity

$\vec{v}$ = the velocity vector

$\hat{n}$ = a unit vector normal to the surface,

S = the inlet or outlet

$$\overline{r}=\frac{1}{S}\int_S rdS$$

31.10.2.1.3. Average Total Pressure

The area-averaged total pressure is defined as follows:

$$\overline{p}_t=\frac{\int_A p_t dA}{A} \tag{31.4}$$

where p_t is the total pressure and A is the area of the inlet or outlet.

The mass-averaged total pressure is defined as follows:

$$\overline{p}_t=\frac{\int_A \left(\rho p_t|\vec{v}\cdot\hat{n}|\right)dA}{\int_A \left(\rho|\vec{v}\cdot\hat{n}|\right)dA} \tag{31.5}$$

where,

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

p_t = the total pressure

A = the area of the inlet or outlet

$\vec{v}$ = the velocity vector

ρ = the fluid density

$\hat{n}$ = a unit vector normal to the surface

31.10.2.1.4. Average Total Temperature

The area-averaged total temperature is defined as follows:

$$\overline{T}_t = \frac{\int_A T_t dA}{A} \tag{31.6}$$

where T_t is the total temperature and A is the area of the inlet or outlet.

The mass-averaged total temperature is defined as follows:

$$\overline{T}_t = \frac{\int_A \left(\rho T_t \left|\vec{v}\cdot\hat{n}\right|\right) dA}{\int_A \left(\rho \left|\vec{v}\cdot\hat{n}\right|\right) dA} \tag{31.7}$$

where,

T_t = the total temperature

A = the area of the inlet or outlet

$\vec{v}$ = the velocity vector

ρ = the fluid density

$\hat{n}$ = a unit vector normal to the surface

31.10.2.1.5. Average Flow Angles

The area-averaged flow angles are defined as follows:

$$\overline{\alpha}_r = \tan^{-1}\left(\frac{\int_A v_\theta dA}{\int_A v_z dA}\right) \tag{31.8}$$

in the radial direction, and

$$\overline{\alpha}_\theta = \tan^{-1}\left(\frac{\int_A v_r dA}{\int_A v_z dA}\right) \tag{31.9}$$

in the tangential direction, where v_z, v_r, and v_θ represent the axial, radial, and tangential velocities, respectively.

The mass-averaged flow angles are defined as follows:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\overline{\alpha}_{r,m} = \tan^{-1}\left(\frac{\int_A (\rho v_r) dA}{\int_A (\rho v_z) dA}\right) \tag{31.10}$$

in the radial direction, and

$$\overline{\alpha}_{\theta,m} = \tan^{-1}\left(\frac{\int_A (\rho v_\theta) dA}{\int_A (\rho v_z) dA}\right) \tag{31.11}$$

in the tangential direction.

31.10.2.1.6. Passage Loss Coefficient

The engineering loss coefficient is defined as follows:

$$K_L = \frac{\overline{p}_{t,i} - \overline{p}_{t,o}}{\frac{1}{2}\rho \overline{v}_i^2} \tag{31.12}$$

where,

$\overline{p}_{t,i}$ = the mass-averaged total pressure at the inlet

$\overline{p}_{t,o}$ = the mass-averaged total pressure at the outlet

ρ = the density of the fluid

$\overline{v}_i$ = the mass-averaged velocity magnitude at the inlet

The normalized loss coefficient is defined as follows:

$$K_{L,n} = \frac{\overline{p}_{t,i} - \overline{p}_{t,o}}{\overline{p}_{t,i} - \overline{p}_{s,o}} \tag{31.13}$$

where $\overline{p}_{s,o}$ is the mass-averaged static pressure at the outlet.

31.10.2.1.7. Axial Force

The axial force on the rotating parts is defined as follows:

$$F_a = \left(\int_S \left(\overline{\overline{\tau}} \cdot \hat{n}\right) dS\right) \cdot \hat{a} \tag{31.14}$$

where,

S = the surfaces comprising all rotating parts

$\overline{\overline{\tau}}$ = the total stress tensor (pressure and viscous stresses)

$\hat{n}$ = a unit vector normal to the surface

$\hat{a}$ = a unit vector parallel to the axis of rotation

31.10.2.1.8. Torque

The torque on the rotating parts is defined as follows:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$T=\left(\int_S \left(\vec{r}\times\left(\bar{\bar{\tau}}\cdot\hat{n}\right)\right) dS\right)\cdot\hat{a} \tag{31.15}$$

where,

S = the surfaces comprising all rotating parts

$\bar{\bar{\tau}}$ = the total stress tensor

$\hat{n}$ = a unit vector normal to the surface

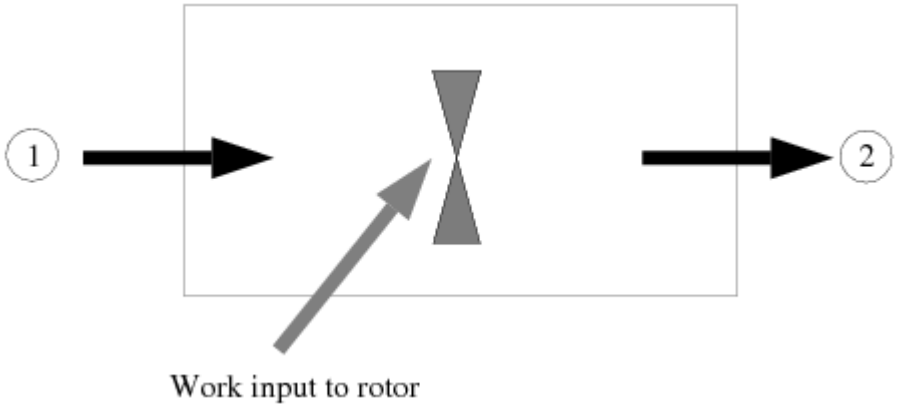
$\vec{r}$ = the position vector

$\hat{a}$ = a unit vector parallel to the axis of rotation

31.10.2.1.9. Efficiencies for Pumps and Compressors

The definitions of the efficiencies for compressible and incompressible flows in pumps and compressors are described in this section. Efficiencies for turbines are described later in this section. Consider a pumping or compression device operating between states 1 and 2 as illustrated in Figure 31.82: Pump or Compressor (p. 1864). Work input to the device is required to achieve a specified compression of the working fluid.

Figure 31.82: Pump or Compressor



Assuming that the processes are steady state, steady flow, and that the mass flow rates are equal at the inlet and outlet of the device (no film cooling, bleed air removal, and so on), the efficiencies for incompressible and compressible flows are as described in the following subsections.

31.10.2.1.9.1. Incompressible Flows

For devices such as liquid pumps and fans (2D boundaries or 3D cell zones) at low speeds, the working fluid can be treated as incompressible. The efficiency of a pumping process with an incompressible working fluid is defined as the ratio of the head rise achieved by the fluid to the power supplied to the rotor/impeller. This can be expressed as follows:

$$\eta=\frac{Q\left(p_{t2}-p_{t1}\right)}{T\omega} \tag{31.16}$$

where,

Q = volumetric flow rate

p_t = total pressure

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

T = net torque acting on the rotor/impeller

ω = rotational speed

This definition is sometimes called the "hydraulic efficiency". Often, other efficiencies are included to account for flow leakage (volumetric efficiency) and mechanical losses along the transmission system between the rotor and the machine providing the power for the rotor/impeller (mechanical efficiency). Incorporating these losses then yields a total efficiency for the system.

31.10.2.1.9.2. Compressible Flows

For gas compressors that operate at high speeds and high pressure ratios, the compressibility of the working fluid must be taken into account. The efficiency of a compression process with a compressible working fluid is defined as the ratio of the work required for an ideal (reversible) compression process to the actual work input. This assumes the compression process occurs between states 1 and 2 *for a given pressure ratio*. In most cases, the pressure ratio is the *total pressure at state 2* divided by the *total pressure at state 1*. If the process is also adiabatic, then the ideal state at 2 is the *isentropic* state.

From the foregoing definition, the efficiency for an adiabatic compression process can be written as

$$\eta_c = \frac{h_{t2,i} - h_{t1}}{h_{t2} - h_{t1}} \tag{31.17}$$

where,

h_{t1} = total enthalpy at 1

h_{t2} = actual total enthalpy at 2

$h_{t2,i}$ = isentropic total enthalpy at 2

If the specific heat is constant, Equation 31.17 (p. 1865) can also be expressed as

$$\eta_c = \frac{T_{t2,i} - T_{t1}}{T_{t2} - T_{t1}} \tag{31.18}$$

where,

T_{t1} = total temperature at 1

T_{t2} = actual total temperature at 2

$T_{t2,i}$ = isentropic total temperature at 2

Using the isentropic relation

$$\frac{T_{t2,i}}{T_{t1}} = \left(\frac{p_{t2}}{p_{t1}}\right)^{\frac{\gamma-1}{\gamma}} \tag{31.19}$$

where γ is the ratio of specific heats specified in the Reference Values Task Page (p. 2502).

The efficiency can be written in the compact form

$$\eta_c = \frac{T_{t1}\left[\left(\frac{p_{t2}}{p_{t1}}\right)^{\frac{\gamma-1}{\gamma}} - 1\right]}{T_{t2} - T_{t1}} \tag{31.20}$$

Note that this definition requires data only for the *actual* states 1 and 2.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

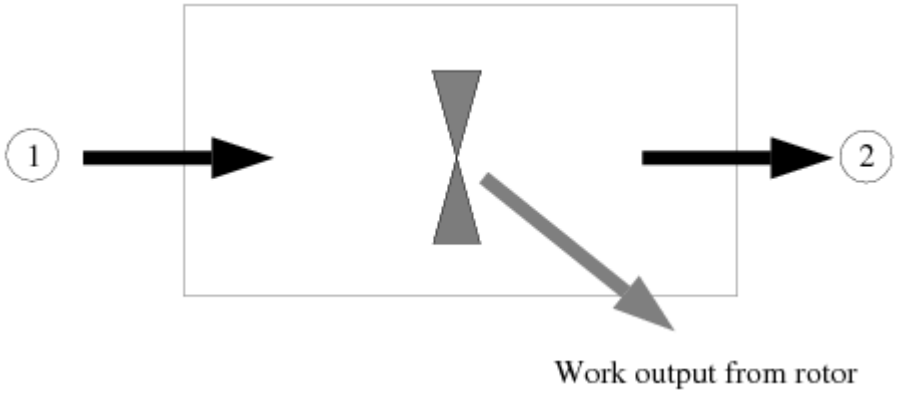
Compressor designers also make use of the polytropic efficiency when comparing one compressor with another. The polytropic efficiency is defined as follows:

$$\eta_{c,p}=\frac{\frac{\gamma-1}{\gamma}\ln\left(\frac{p_{t2}}{p_{t1}}\right)}{\ln\left(\frac{T_{t2}}{T_{t1}}\right)} \tag{31.21}$$

31.10.2.1.10. Efficiencies for Turbines

Consider a turbine operating between states 1 and 2 in Figure 31.83: Turbine (p. 1866). Work is extracted from the working fluid as it expands through the turbine. Assuming that the processes are steady state, steady flow, and that the mass flow rates are equal at the inlet and outlet of the device (no film cooling, bleed air removal, and so on), turbine efficiencies for incompressible and compressible flows are as described below.

Figure 31.83: Turbine



31.10.2.1.10.1. Incompressible Flows

The efficiency of a turbine with an incompressible working fluid is defined as the ratio of the work delivered to the rotor to the energy available from the fluid stream. This ratio can be expressed as follows:

$$\eta=\frac{T\omega}{Q\left(p_{t1}-p_{t2}\right)} \tag{31.22}$$

where,

Q = volumetric flow rate

p_t = total pressure

T = net torque acting on the rotor/impeller

ω = rotational speed

Note the similarity between this definition and the definition of incompressible compression efficiency (Equation 31.16 (p. 1864)). As with hydraulic pumps and compressors, other efficiencies (for example, volumetric and mechanical efficiencies) can be defined to account for other losses in the system.

31.10.2.1.10.2. Compressible Flows

For high-speed gas turbines operating at large expansion pressure ratios, compressibility must be accounted for. The efficiency of an expansion process with a compressible working fluid is defined as the ratio of the actual work extracted from the fluid to the work extracted from an ideal (reversible) process.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This assumes that the expansion process occurs between states 1 and 2 *for a given pressure ratio*. In contrast to the compression process, the pressure ratio for expansion is the *total pressure at state 1* divided by the *total pressure at state 2*. If the process is also adiabatic, then the ideal state at 2 is the *isentropic* state.

From the foregoing definition, the efficiency for an adiabatic expansion process through a turbine can be written as

$$\eta_c = \frac{h_{t1} - h_{t2}}{h_{t1} - h_{t2,i}} \tag{31.23}$$

where

h_{t1} = total enthalpy at 1

h_{t2} = actual total enthalpy at 2

$h_{t2,i}$ = isentropic total enthalpy at 2

If the specific heat is constant, Equation 31.23 (p. 1867) can also be expressed as

$$\eta_e = \frac{T_{t1} - T_{t2}}{T_{t1} - T_{t2,i}} \tag{31.24}$$

where

T_{t1} = total temperature at 1

T_{t2} = actual total temperature at 2

$T_{t2,i}$ = isentropic total temperature at 2

Using the isentropic relation

$$\frac{T_{t1}}{T_{t2,i}} = \left(\frac{p_{t1}}{p_{t2}}\right)^{\frac{\gamma-1}{\gamma}} \tag{31.25}$$

the expansion efficiency can be written in the compact form

$$\eta_e = \frac{T_{t1} - T_{t2}}{T_{t1}\left[1 - \left(\frac{p_{t2}}{p_{t1}}\right)^{\frac{\gamma-1}{\gamma}}\right]} \tag{31.26}$$

Note that this definition requires data only for the *actual* states 1 and 2.

As with compressors, one may also define a polytropic efficiency for turbines. The polytropic efficiency is defined as follows:

$$\eta_{e,p} = \frac{\ln\left(\frac{T_{t1}}{T_{t2}}\right)}{\frac{\gamma-1}{\gamma}\ln\left(\frac{p_{t1}}{p_{t2}}\right)} \tag{31.27}$$

31.10.3. Displaying Turbomachinery Averaged Contours

Turbo averaged contours are generated as projections of the values of a variable averaged in the circumferential direction and visualized on an r- z plane. A sample plot is shown in Figure 31.85: Turbo Averaged Filled Contours of Static Pressure (p. 1869).

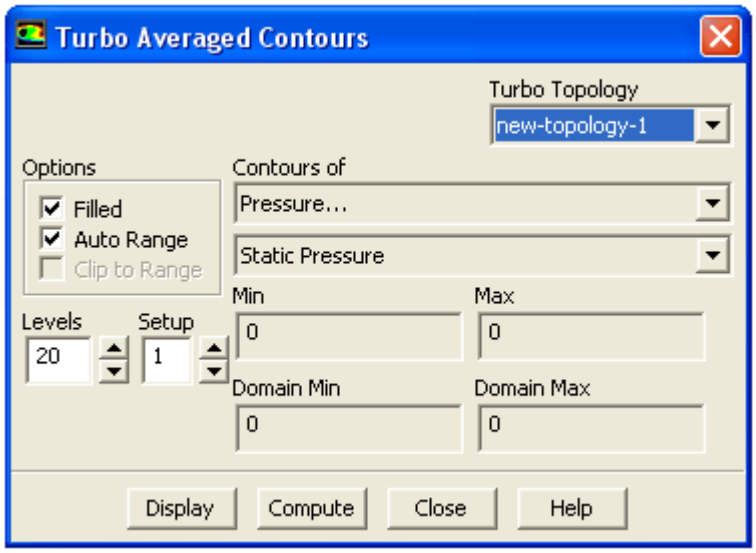
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.10.3.1. Steps for Generating Turbomachinery Averaged Contour Plots

You can display contours using the Turbo Averaged Contours Dialog Box (p. 2816) (Figure 31.84: The Turbo Averaged Contours Dialog Box (p. 1868)).

Postprocessing → Model Specific → Turbo Topology → Averaged Contours...

Figure 31.84: The Turbo Averaged Contours Dialog Box



The basic steps for generating a turbo averaged contour plot are as follows:

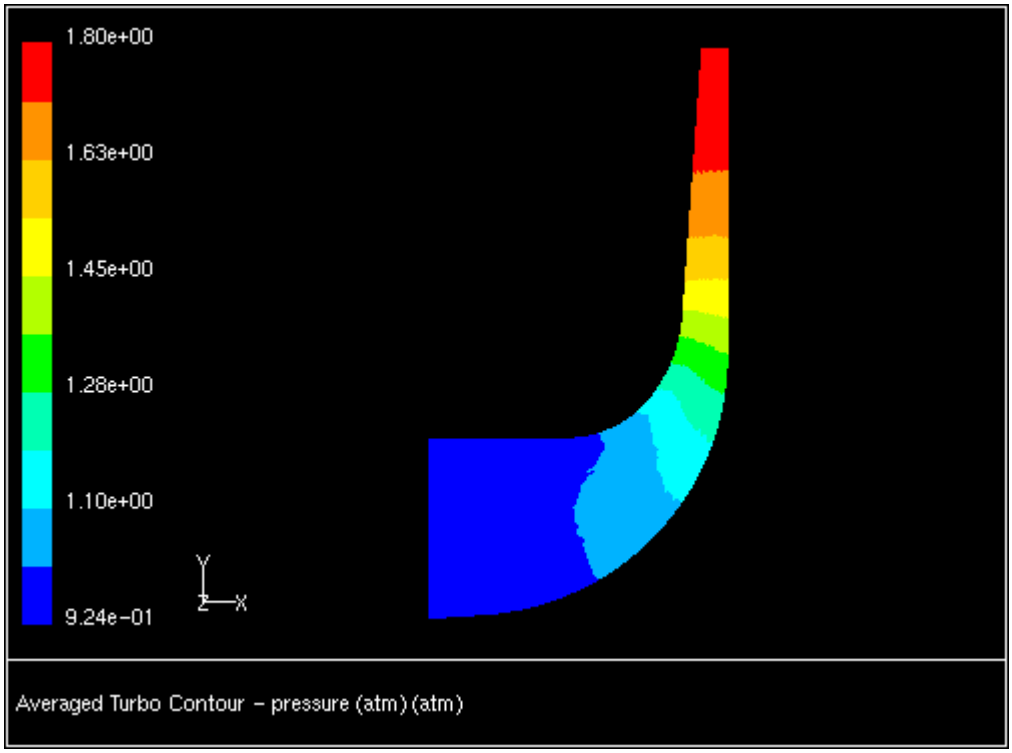
1. Select **All** or a specific predefined turbomachinery topology from the **Turbo Topology** drop-down list.
2. Select the variable or function to be displayed in the **Contours of** drop-down list. First select the desired category in the upper list; you may then select a related quantity in the lower list. (See Turbomachinery-Specific Variables (p. 1872) for a list of turbomachinery-specific variables, and see Field Function Definitions (p. 1931) for an explanation of the variables in the list.)
3. Specify the number of contours in the **Levels** field. The maximum number of levels allowed is 100.
4. Set any of the options described below.
5. Click **Display** to draw the specified contours in the active graphics window.

The resulting display will include the specified number of contours of the selected variable, with the magnitude on each one determined by equally incrementing between the values shown in the **Min** and **Max** fields.

Note that the **Min** and **Max** values displayed in the dialog box are the minimum and maximum averaged values. These limits will in general be different from the global **Domain Min** and **Domain Max**, which are also displayed for your reference (see Figure 31.84: The Turbo Averaged Contours Dialog Box (p. 1868)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.85: Turbo Averaged Filled Contours of Static Pressure



31.10.3.2. Contour Plot Options

The options mentioned in the procedure above include drawing color-filled contours (instead of line contours), specifying a range of values to be contoured, and storing the contour plot settings. These options are the same as those in the standard Contours Dialog Box (p. 2676). See Contour and Profile Plot Options (p. 1759) for details about using them.

31.10.4. Displaying Turbomachinery 2D Contours

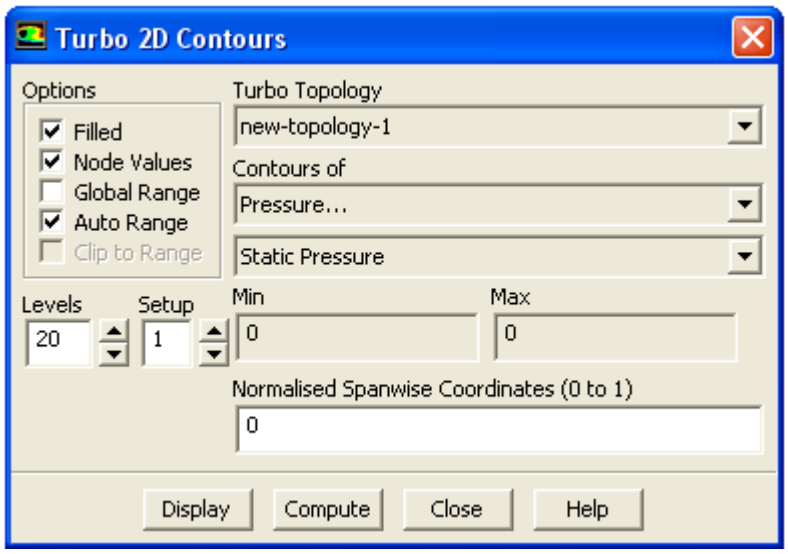
In postprocessing a turbomachinery solution, it is often desirable to display contours on surfaces of constant spanwise coordinate, and then project these contours onto a plane. This permits easier evaluation of the contours, especially for surfaces that are highly three-dimensional.

31.10.4.1. Steps for Generating Turbo 2D Contour Plots

You can display contours using the Turbo 2D Contours Dialog Box (p. 2814) (Figure 31.86: The Turbo 2D Contours Dialog Box (p. 1870)).

Postprocessing → Model Specific → Turbo Topology → 2D Contours...

Figure 31.86: The Turbo 2D Contours Dialog Box



The basic steps for generating a turbo 2D contour plot are as follows:

1. Specify a specific predefined turbomachinery topology using the **Turbo Topology** drop-down list.

2. Enter a value for the **Normalized Spanwise Coordinates (0 to 1)** for the spanwise surface you want to create.

   Important

   If shroud and hub are the curved surfaces, the isosurface very close to them may contain void spaces as ANSYS Fluent displays only a plane cut surface.

3. Select the variable or function to be displayed in the **Contours of** drop-down list.

   First select the desired category in the upper list; you may then select a related quantity in the lower list. See Turbomachinery-Specific Variables (p. 1872) for a list of turbomachinery-specific variables, and see Field Function Definitions (p. 1931) for an explanation of the variables in the list.

4. Specify the number of contours in the **Levels** field. The maximum number of levels allowed is 100.

5. Click **Display** to draw the specified contours in the active graphics window.

The resulting display will include the specified number of contours of the selected variable, with the magnitude on each one determined by equally incrementing between the values shown in the **Min** and **Max** fields.

31.10.4.2. Contour Plot Options

Depending on the type of contour plot you want to display, select appropriate choice under **Options**. These options are the same as those in the standard Contours Dialog Box (p. 2676). See Contour and Profile Plot Options (p. 1759) for details about using them.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.10.5. Generating Averaged XY Plots of Turbomachinery Solution Data

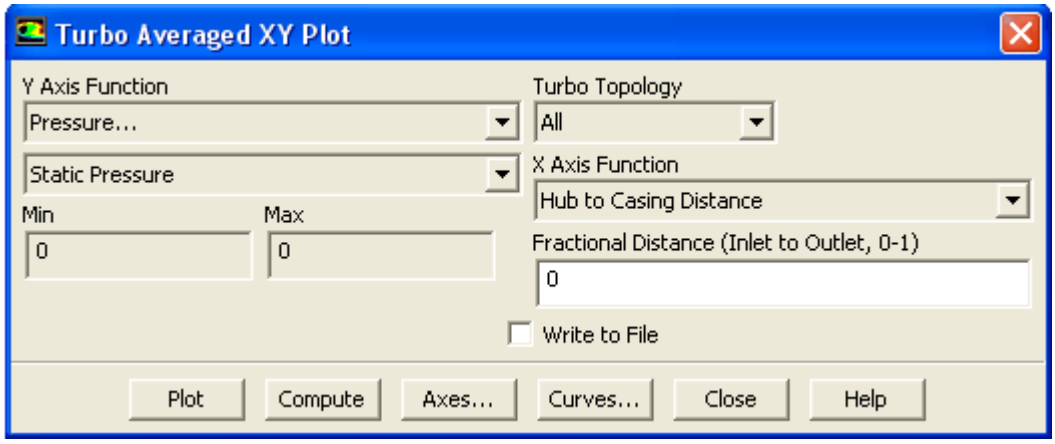
When comparing numerical solutions of turbomachinery problems to experimental data, it is often useful to plot circumferentially averaged quantities in the spanwise and meridional directions. This section describes how to do this in ANSYS Fluent.

31.10.5.1. Steps for Generating Turbo Averaged XY Plots

To create an XY plot of circumferentially averaged solution data, you will use the Turbo Averaged XY Plot Dialog Box (p. 2817) (Figure 31.87: The Turbo Averaged XY Plot Dialog Box (p. 1871)).

Postprocessing → Model Specific → Turbo Topology → Averaged XY Plot...

Figure 31.87: The Turbo Averaged XY Plot Dialog Box



The procedure for generating a turbo averaged XY plot are as follows:

1. Select the variable or function to be plotted in the **Y Axis Function** drop-down list. First select the desired category in the upper list; you may then select a related quantity in the lower list. (See Turbomachinery-Specific Variables (p. 1872) for a list of turbomachinery-specific variables, and see Field Function Definitions (p. 1931) for an explanation of the variables in the list.)

2. Select **All** or a specific predefined turbomachinery topology from the **Turbo Topology** drop-down list.

3. Select the variable or function to be plotted in the **X Axis Function** drop-down list. The choices are **Hub to Casing Distance** and **Meridional Distance**.

4. Specify the desired value in the **Fractional Distance** field. The definition of the fractional distance depends on your selection of **X Axis Function**:

   - If you select **Hub to Casing Distance**, the fractional distance will be **Inlet to Outlet**.
   - If you select **Meridional Distance**, the fractional distance will be **Hub to Casing**.

5. (optional) Modify the attributes of the axes or curves as described in Modifying Axis Attributes (p. 1852) and Modifying Curve Attributes (p. 1854).

6. Click **Plot** to generate the XY plot in the active graphics window.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that you can use any of the mouse buttons to annotate the XY plot (see Adding Text to the Graphics Window (p. 1789)).

If you want to write the XY data to a file, follow these steps instead of Step 5 above:

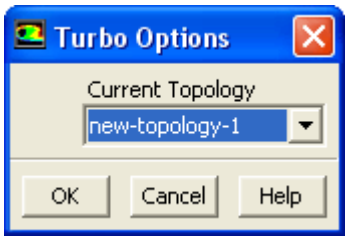
1. Enable the **Write to File** option. The **Plot** button changes to a **Write...** button.
2. Click **Write...**.
3. In The Select File Dialog Box (p. 19), specify a name for the plot file and save it.

31.10.6. Globally Setting the Turbomachinery Topology

In some cases, that is, isosurface creation, ANSYS Fluent enables you to globally set the current turbomachinery topology for your model using the Turbo Options Dialog Box (p. 2819) (Figure 31.88: The Turbo Options Dialog Box (p. 1872)).

Postprocessing → Model Specific → Turbo Topology → Options...

Figure 31.88: The Turbo Options Dialog Box



To set the current topology, select a topology from the **Current Topology** drop-down list and select **OK**.

31.10.7. Turbomachinery-Specific Variables

The following turbomachinery-specific variables are available in ANSYS Fluent:

- **Meridional Coordinate**
- **Abs Meridional Coordinate**
- **Spanwise Coordinate**
- **Abs (H-C) Spanwise Coordinate**
- **Abs (C-H) Spanwise Coordinate**
- **Pitchwise Coordinate**
- **Abs Pitchwise Coordinate**

These variables are contained in the **Mesh...** category of the variable selection drop-down list. See Field Function Definitions (p. 1931) for their definitions.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.11. Fast Fourier Transform (FFT) Postprocessing

When interpreting time-sequence data from a transient solution, it is often useful to look at the data's spectral (frequency) attributes. For instance, you may want to determine the major vortex-shedding frequency from the time-history of the drag force on a body recorded during an ANSYS Fluent simulation. Or, you may want to compute the spectral distribution of static pressure data recorded at a particular location on a body surface. Similarly, you may need to compute the spectral distribution of turbulent kinetic energy using data for fluctuating velocity components. To interpret some of these time dependent data, you need to perform Fourier transform analysis. In essence, the Fourier transform enables you to take any time dependent data and resolve it into an equivalent summation of sine and cosine waves.

ANSYS Fluent enables you to analyze your time dependent data using the Fast Fourier Transform (FFT) algorithm. Information on using the FFT algorithm in ANSYS Fluent is provided in the following sections:

31.11.1. Limitations of the FFT Algorithm
31.11.2. Windowing
31.11.3. Fast Fourier Transform (FFT)
31.11.4. Using the FFT Utility

31.11.1. Limitations of the FFT Algorithm

The following limitations apply to ANSYS Fluent's FFT module:

- The ANSYS Fluent FFT module can only read inputs files in the ANSYS Fluent report file, monitor, and x-y file formats.
- The ANSYS Fluent FFT module assumes that the input data have been sampled at equal intervals and are consecutive (in the order of increasing time).
- The lowest frequency that the FFT module can pick up is given by $1/t$, where t is the total sampling time. If the sampled sequence contains frequencies lower than this, these frequencies will be aliased into higher frequencies.
- The highest frequency that the FFT module can pick up is $1/\left(2dt\right)$, where dt is the sampling interval (or time step).

31.11.2. Windowing

The discrete FFT algorithm is based on the assumption that the time-sequence data passed to the FFT corresponds to a single period of a periodically repeating signal. Since, in most situations, the first and the last data points will not coincide, the repeating signal implied in the assumption can often have a large discontinuity. The large discontinuity produces high-frequency components in the resulting Fourier modes, causing an aliasing error. You can condition the input signal before the transform by "windowing" it, in order to avoid this problem.

Suppose that we have N consecutive discrete (time-sequence) data sampled with a constant interval, Δt:

$$\phi_k \equiv \phi\left(t_k\right), \quad t_k \equiv k\Delta t, \qquad k=0,1,2,\ldots,\left(N-1\right) \tag{31.28}$$

Windowing is done by multiplying the original input data (ϕ_j) by a window function, W_j:

$$\tilde{\phi}_j = \phi_j W_j \qquad j=0,1,2,\ldots,\left(N-1\right) \tag{31.29}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS Fluent offers four different window functions:

Hamming's window:

$$W_j = \begin{cases} 0.54 - 0.46\cos\left(\frac{8\pi j}{N}\right) & j \le \frac{N}{8},\ j \ge \frac{7N}{8} \\ 1 & \frac{N}{8} < j < \frac{7N}{8} \end{cases} \tag{31.30}$$

Hanning's window:

$$W_j = \begin{cases} 0.5\left[1 - \cos\left(\frac{8\pi j}{N}\right)\right] & j \le \frac{N}{8},\ j \ge \frac{7N}{8} \\ 1 & \frac{N}{8} < j < \frac{7N}{8} \end{cases} \tag{31.31}$$

Barlett's window:

$$W_j = \begin{cases} \frac{8j}{N} & j \le \frac{N}{8} \\ 8\left(1 - \frac{j}{N}\right) & j \ge \frac{7N}{8} \\ 1 & \frac{N}{8} < j < \frac{7N}{8} \end{cases} \tag{31.32}$$

Blackman's window:

$$W_j = \begin{cases} 0.42 - 0.5\cos\left(\frac{8\pi j}{N}\right) + 0.08\cos\left(\frac{16\pi j}{N}\right) & j \le \frac{N}{8},\ j \ge \frac{7N}{8} \\ 1 & \frac{N}{8} < j < \frac{7N}{8} \end{cases} \tag{31.33}$$

These window functions preserve a large fraction $(3/4)$ of the original data, affecting only $1/4$ of the data on both ends.

31.11.3. Fast Fourier Transform (FFT)

The Fourier transform utility in ANSYS Fluent enables you to compute the Fourier transform of a signal, $\phi(t)$, a real-valued function, from a finite number of its sampled points.

For a periodic set of N sampled points, ϕ_k, the discrete Fourier transform [13] (p. 2887) expresses the signal as a finite trigonometric series:

$$\phi_k = \sum_{n=0}^{N-1} \hat{\phi}_n e^{2\pi i k n / N} \qquad k = 0, 1, 2, \ldots (N-1) \tag{31.34}$$

where the series coefficients $\hat{\phi}_n$ are computed as

$$\hat{\phi}_n = \frac{1}{N} \sum_{k=0}^{N-1} \phi_k e^{-2\pi i k n / N} \qquad n = 0, 1, 2, \ldots (N-1) \tag{31.35}$$

Equation 31.34 (p. 1874) and Equation 31.35 (p. 1874) form a Fourier transform pair that enables us to determine one from the other.

Note that when we follow the convention of varying n from 0 to $N-1$ in Equation 31.34 (p. 1874) and Equation 31.35 (p. 1874) instead of from $-N/2$ to $N/2$, the range of index $1 \le n \le N/2 - 1$ corresponds to positive frequencies, and the range of index $N/2 + 1 \le n \le N-1$ corresponds to negative frequencies. $n = 0$ still corresponds to zero frequency.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For the actual calculation of the transforms, ANSYS Fluent adopts the so-called fast Fourier transform (FFT) algorithm, which significantly reduces operation counts in comparison to the direct transform. Furthermore, unlike most FFT algorithms in which the number of data should be a power of 2, the FFT utility in ANSYS Fluent employs a prime-factor algorithm [121] (p. 2893). The number of data points permissible in the prime-factor FFT algorithm is any products of mutually prime factors from the set $2,3,4,5,7,8,9,11,13,16$, with a maximum value of $720720=5\times7\times9\times11\times13\times16$. Thus, the prime-factor FFT preserves the original data better than the conventional FFT.

Just prior to computing the transform, ANSYS Fluent determines the largest permissible number of data points based on the prime factors, discarding the rest of the data. A list of these numbers is provided in Table 31.2: Numbers of Data Points Supported by the Prime-Factor FFT Algorithm (p. 1875).

Table 31.2: Numbers of Data Points Supported by the Prime-Factor FFT Algorithm

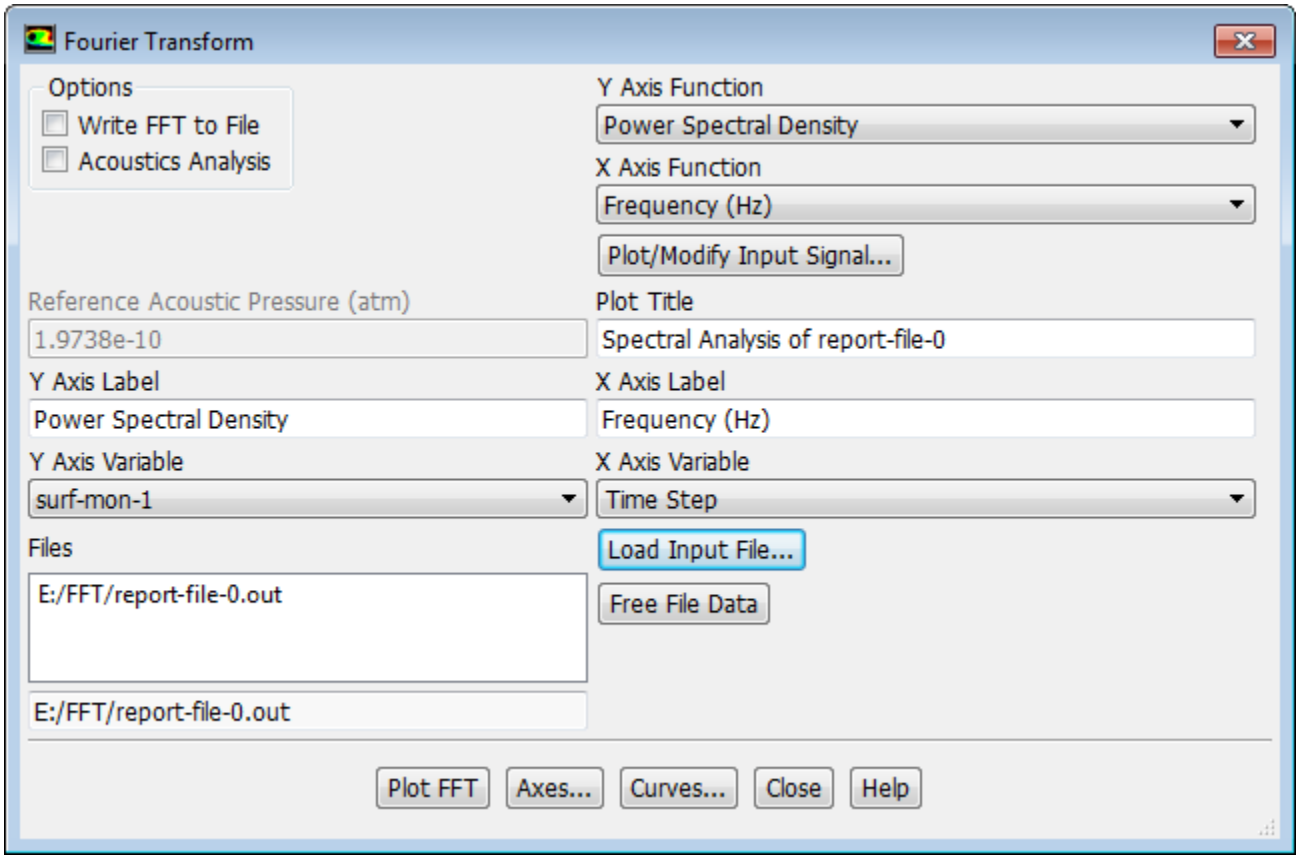
2	70	220	572	1386	3120	8580	34320
4	72	234	616	1430	3276	9240	36036
6	78	240	624	1456	3432	9360	40040
8	80	252	630	1540	3640	10010	48048
10	84	260	660	1560	3696	10296	51480
12	88	264	720	1584	3960	10920	55440
14	90	280	728	1638	4004	11088	60060
16	104	286	770	1680	4290	11440	65520
18	110	308	780	1716	4368	12012	72072
20	112	312	792	1820	4620	12870	80080
22	120	330	840	1848	4680	13104	90090
24	126	336	858	1872	5040	13860	102960
26	130	360	880	1980	5148	16016	120120
28	132	364	910	2002	5460	16380	144144
30	140	390	924	2184	5544	17160	180180
36	144	396	936	2288	5720	18018	240240
39	154	420	990	2310	6006	18480	360360
40	156	440	1008	2340	6160	20020	720720
42	168	462	1040	2520	6552	20592	
44	176	468	1092	2574	6864	21840	
48	180	504	1144	2640	6930	24024	
52	182	520	1170	2730	7280	25740	
56	198	528	1232	2772	7920	27720	
60	208	546	1260	2860	8008	30030	
66	210	560	1320	3080	8190	32760	

31.11.4. Using the FFT Utility

The ANSYS Fluent FFT utility is available through the Fourier Transform Dialog Box (p. 2602) (Figure 31.89: The Fourier Transform Dialog Box (p. 1876)).

Results → Plots → FFT → Set Up...

Figure 31.89: The Fourier Transform Dialog Box



31.11.4.1. Loading Data for Spectral Analysis

FFT analysis requires an input signal data file consisting of time-sequence data. To load an input signal data file into the Fourier Transform Dialog Box (p. 2602), click **Load Input File...**. This displays The Select File Dialog Box (p. 19) where you can browse through your file directories and locate your data file containing your time-sequence data. To remove a file from the **Files** list, select it and then click **Free File Data**.

If you computed acoustic signals "on the fly", you have the option of processing signal data from a file or processing receiver data stored in memory. To analyze signal data from an existing input file, select **Process File Data** under **Process Options** and proceed as described above. To analyze receiver data stored in memory, select **Process Receiver** under **Process Options** and select the appropriate receiver in the **Receiver** list.

Click **Plot FFT** to display the spectral analysis data and, if you have enabled **Acoustics Analysis**, to calculate the overall sound pressure level in dB based on the **Reference Acoustic Pressure** and display it in the console.

Note

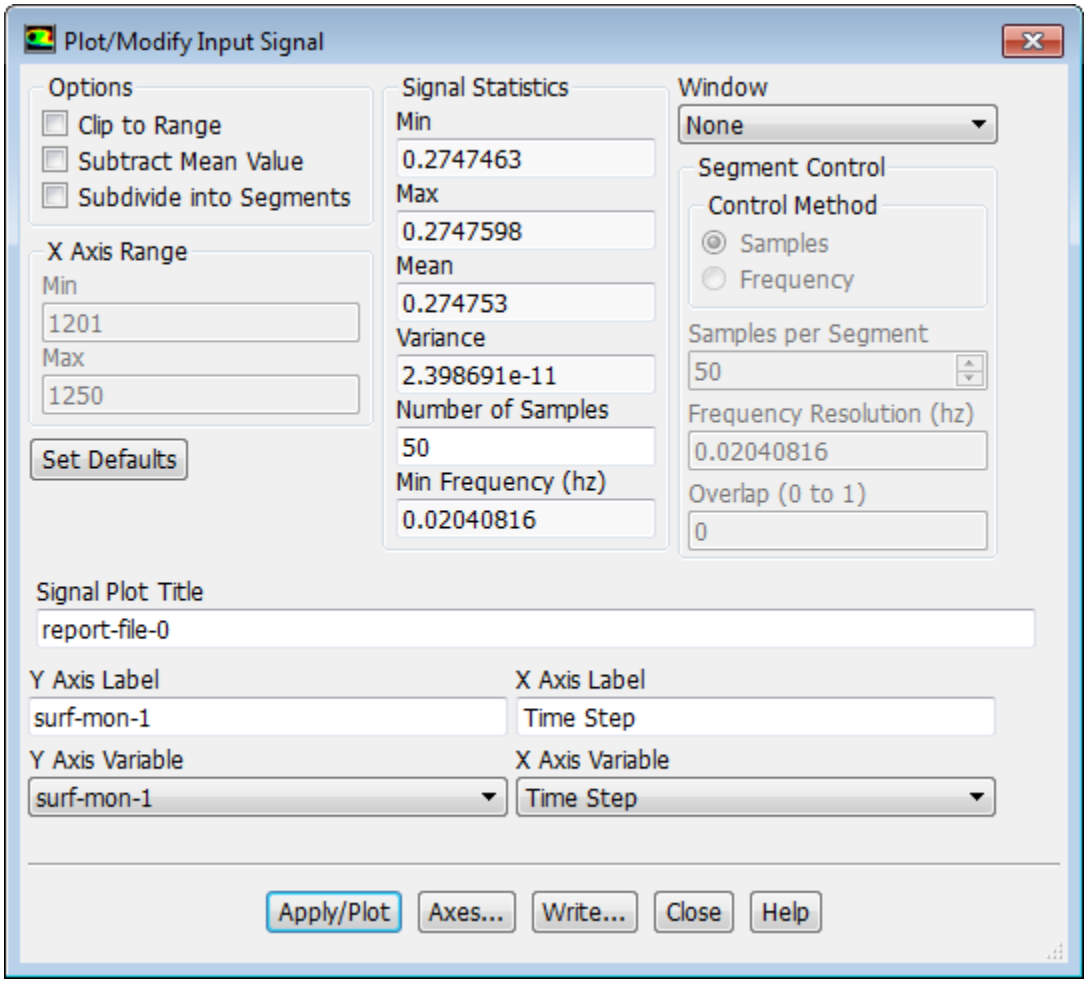
You can specify the y axis and x axis variables if your input report file contains multiple report definitions.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.11.4.2. Customizing the Input and Defining the Spectrum Smoothing

With the input signal data file loaded into the Fourier Transform Dialog Box (p. 2602), you may want to view a plot of the input signal and/or customize the data set in preparation for applying the FFT algorithm. To do so, click the **Plot/Modify Input Signal** button to open the Plot/Modify Input Signal Dialog Box (p. 2605) (Figure 31.90: The Plot/Modify Input Signal Dialog Box (p. 1877)).

Figure 31.90: The Plot/Modify Input Signal Dialog Box



The Plot/Modify Input Signal Dialog Box (p. 2605) allows you to analyze a portion of the input signal, view input **Signal Statistics**—**Min**, **Max**, **Mean**, **Variance**, total **Number of Samples**, and **Min Frequency** (that is, the finest possible frequency resolution)—and set title and label information for the input signal plot. Additionally, this dialog box allows you to enable and control the smoothing of the resulting spectrum by subdividing the input signal into multiple segments and averaging the segment-based spectra.

Note

You can specify the y axis and x axis variables if your input report file contains multiple report definitions.

31.11.4.2.1. Customizing the Input Signal Data Set

By default, the entire data set is analyzed. To analyze a portion of the input signal, enable the **Clip to Range** option and specify the data range by entering **Min** and **Max** values under **X Axis Range**. To have the y-axis quantities reduced by the **Mean** value of the relevant signal property, enable the **Subtract Mean Value** option.

The **Set Defaults** button will reset the original values for the **Min** and **Max** fields under **X Axis Range** and turn off the **Clip to Range** option.

31.11.4.2.2. Spectrum Smoothing Through Signal Segmentation

For long broadband noise signals, the computed Fourier spectrum may display spurious fluctuations of amplitudes between the neighboring modes. In order to obtain a smooth spectrum, you can split the signal into multiple overlapping segments so that ANSYS Fluent can apply the FFT algorithm on each segment and then average the resulting spectra. This procedure allows you to significantly suppress the spurious fluctuations at the cost of coarsening the frequency resolution of the spectrum. Note that the arithmetic averaging of spectra is performed for the squares of Fourier amplitudes, thus conserving the signal energy.

To use the smoothing procedure, enable the **Subdivide into Segments** option and define the segment size and the overlap of the subsequent segments in the **Segment Control** group box. Make a selection from the **Control Method** list to specify how you want to define the segment size: select **Samples** if you want to specify the number of **Samples per Segment**; select **Frequency** if you want to specify the desired **Frequency Resolution** Δf in Hertz units (the segment length in seconds is then equal to $1/\Delta f$). To help determine a suitable segment size, you can use the **Number of Samples** and the **Min Frequency** information in the **Signal Statistics** group box. Note that the final segment length is selected by ANSYS Fluent to adhere to the largest supported number of data samples (see Table 31.2: Numbers of Data Points Supported by the Prime-Factor FFT Algorithm (p. 1875)). The actual signal length and the number of segments will be displayed in the console when you plot or write the Fourier spectrum. The number of segments depends on the **Overlap** of the subsequent segments, which is specified regardless of which **Control Method** you select. If you select a **Window** function (see Windowing (p. 1873) for details), it is recommended that you use an **Overlap** value of at least 0.125, in order to cover the signal portion affected by the window function. Generally, an **Overlap** value of 0.5 is recommended. The exact number of samples that are in the overlapping region of the segment can be determined from the segment hop size, which is displayed in the console. The segment hop size is the distance (in terms of samples) between the starting samples of any two adjacent segments (that is, the shift from one segment to the next), and has a minimum value of 1.

The **Set Defaults** button will reset the segment control values and disable the **Subdivide into Segments** option.

31.11.4.2.3. Viewing Data Statistics

To aid in the signal analysis, whether for the entire input signal or for a certain range of data, the **Signal Statistics** group box in the Plot/Modify Input Signal Dialog Box (p. 2605) displays signal information such as the minimum, maximum, and average signal values, as well as the signal variance, total number of samples, and finest possible frequency resolution.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.11.4.2.4. Customizing Titles and Labels

You can create a new title or edit the original title for the input signal plot by entering a text string in **Signal Plot Title**. Likewise, you can create a new axis label or edit the original axis label by entering a text string into either **Y Axis Label** or **X Axis Label**.

31.11.4.2.5. Applying the Changes in the Input Signal Data

To apply any changes you have made in the Plot/Modify Input Signal Dialog Box (p. 2605) and view a plot of the input signal, click **Apply/Plot**.

31.11.4.3. Customizing the Output

In most practical applications with CFD data, you may want to find out how much power or energy is contained in a certain frequency range, but do not want to distinguish positive and negative frequency. In recognition of this, all the outputs from the FFT module in ANSYS Fluent pertain to *one-sided spectra* for the range of positive frequency.

The Fourier Transform Dialog Box (p. 2602) (Figure 31.89: The Fourier Transform Dialog Box (p. 1876)) and Plot/Modify Input Signal Dialog Box (p. 2605) (Figure 31.90: The Plot/Modify Input Signal Dialog Box (p. 1877)) enable you to set several different functions for the x and y axes, apply different FFT windowing techniques, and set various output options.

31.11.4.3.1. Specifying a Function for the Axis

You can choose the y-axis function using the **Y Axis Function** drop-down list. Available options for the y-axis functions are as follows. Note that the functions related to acoustics (all of which are measured in dB) are only available when the **Acoustics Analysis** option is enabled.

The definitions are provided for $n \le N/2$, and include contributions from both elements of a complex conjugate pair n and N-n. Note that in ANSYS Fluent, the value of N is always an even number.

Power Spectral Density
is the distribution of signal power in the frequency domain. Its value and units depend on the **X Axis Function** choice. For the detailed spectral representation with all resolved harmonics (that is, when **X Axis Function** is either **Frequency**, **Strouhal Number**, or **Fourier Mode**), the **Power Spectral Density** (PSD) has units of the signal magnitude squared over the frequency (for example, Pa^2/Hz) and is defined for the frequency f_n as

$$PSD(f_n) = E(f_n)/\Delta f \qquad n = 1,2,...,N/2 \tag{31.36}$$

where Δf is the frequency step in the discrete spectrum, and the Fourier mode power $E(f_n)$ is computed as

$$E(f_n) = \begin{cases} 0.5\left(2|\hat{\phi}_n|\right)^2 & n = 1,2,...,N/2-1 \\ |\hat{\phi}_n|^2 & n = N/2 \end{cases} \tag{31.37}$$

For the octave analysis (that is, when the **X Axis Function** is either **Octave Band** or **1/3-Octave Band**), the **Power Spectral Density** has units of the signal magnitude squared (for example, Pa^2), and is defined for the frequency band f_{band} as

$$PSD(f_{\text{band}}) = \sum E(f_n) \tag{31.38}$$

where n includes all of the Fourier modes belonging to the band.

Magnitude
is the amplitude. For the detailed spectral representation with all resolved harmonics (that is, when **X Axis Function** is either **Frequency**, **Strouhal Number**, or **Fourier Mode**), the **Magnitude** (A) is defined for the frequency f_n as

$$(f_n) = \begin{cases} |\hat{\phi}_n| & n=0, N/2 \\ 2|\hat{\phi}_n| & n=1,2,...,N/2-1 \end{cases} \tag{31.39}$$

where $A\left(f_0\right)$ is the mean signal value.

For the octave analysis (that is, when the **X Axis Function** is either **Octave Band** or **1/3-Octave Band**), the **Magnitude** is defined for the frequency band f_{band} as

$$A\left(f_{\text{band}}\right) = \sqrt{2PSD\left(f_{\text{band}}\right)} \tag{31.40}$$

where $PSD\left(f_{\text{band}}\right)$ is calculated according to Equation 31.38 (p. 1880).

Sound Pressure Level (dB)
is the decibel level. For either general or acoustic data, when the sampled data is pressure (for example, static pressure or sound pressure), the **Sound Pressure Level (dB)** (L_{sp}) is calculated in decibel units using

$$L_{\text{sp}} = 10\left(\log\frac{PSD}{p_{\text{ref}}^2}\right) \quad (\text{dB}) \tag{31.41}$$

where PSD is the **Power Spectral Density** for either a particular Fourier mode or a particular frequency band (see Equation 31.36 (p. 1879) and Equation 31.38 (p. 1880)). p_{ref} is the reference acoustic pressure, with a default value of 2 × 10^{-5} Pa; you can revise this value, as described in Enabling the FW-H Acoustics Model (p. 1208).

Sound Amplitude (dB)
is similar to the **Sound Pressure Level (dB)**, and is a logarithmic conversion of the pressure signal **Magnitude** into decibel units. The **Sound Amplitude (dB)** (A_{sp}) is calculated for either a Fourier mode or a frequency band using

$$A_{\text{sp}} = 10\left(\log\frac{A}{p_{\text{ref}}}\right) \quad (\text{dB}) \tag{31.42}$$

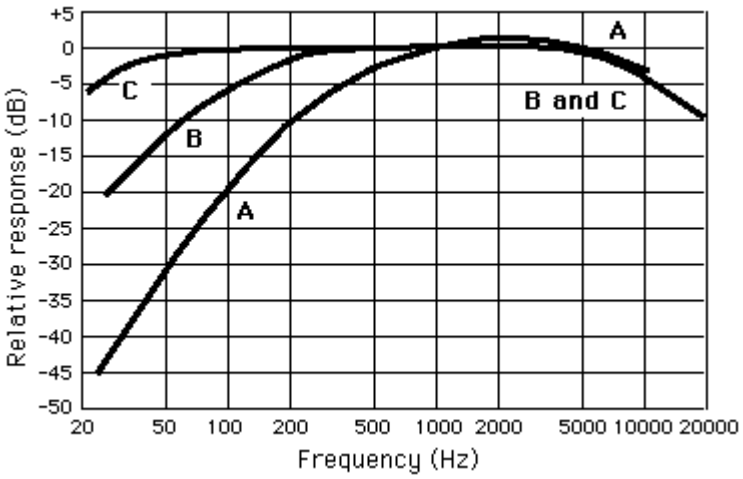
A-Weighted, Sound Pressure Level (dB A)
is the calculated sound pressure level weighted by the A-scale function to more closely approximate the frequency response of the human ear. A-Weighting is applied for loudness levels below 55 phons (55 dB at 1 kHz) and is the most commonly used weighting function. See Figure 31.91: A-, B-, and C-Weighting Functions (p. 1881) for a graphical representation. This function is only available when the **X Axis Function** is either **Octave Band** or **1/3-Octave Band**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

B-Weighted, Sound Pressure Level (dB B)
is the calculated sound pressure level weighted by the B-scale function. B-Weighting is applied to loudness levels between 55 and 85 phons, though it is rarely used. See Figure 31.91: A-, B-, and C-Weighting Functions (p. 1881) for a graphical representation. This function is only available when the **X Axis Function** is either **Octave Band** or **1/3-Octave Band**.

C-Weighted, Sound Pressure Level (dB C)
is the calculated sound pressure level weighted by the C-scale function. C-Weighting is applied for loudness levels above 85 phons and is commonly used for high-intensity sound such as traffic studies. See Figure 31.91: A-, B-, and C-Weighting Functions (p. 1881) for a graphical representation. This function is only available when the **X Axis Function** is either **Octave Band** or **1/3-Octave Band**.

Figure 31.91: A-, B-, and C-Weighting Functions



Further graphical customizations for the y-axis are available by clicking the **Axes...** button. For more information, see Modifying Axis Attributes (p. 1852).

31.11.4.3.2. Specifying a Function for the X Axis

There are three options for the x-axis function you can choose from in order to plot or write the detailed spectrum with all resolved Fourier modes; these three options are related to the discrete frequencies at which the Fourier coefficients are computed. There are also two additional options available when the **Acoustics Analysis** option is enabled, which allow the octave and 1/3 octave band analysis. You can apply specific analytic functions for the x-axis using the **X Axis Function** drop-down list.

Available options for the x-axis functions are as follows. The definitions are provided for $n \le N/2$, because the corresponding definitions for the y-axis functions include contributions from both elements of a complex conjugate pair n and N-n. Note that in ANSYS Fluent, the value of N is always an even number.

Frequency (Hz)
is defined as:

$$f_n = \frac{1}{N\Delta t} n \qquad n = 0, 1, 2, \ldots, N/2 \tag{31.43}$$

where N is the number of data points used in the FFT.

Strouhal Number
is the nondimensionalized version of the frequency defined in Equation 31.43 (p. 1881) :

$$St_n \equiv \frac{f_n L_{ref}}{U_{ref}} \tag{31.44}$$

where L_{ref} and U_{ref} are the reference length and velocity scales specified in the Reference Values Task Page (p. 2502).

Fourier Mode

is the index in Equation 31.34 (p. 1874) and/or Equation 31.35 (p. 1874), which represents the n^{th} term in the Fourier transform of the signal.

Octave Band (Hz)

is a range of discrete frequency bands for different octaves within the threshold of hearing. The range of each octave band is double to that of the previous band (see Table 31.3: Octave Band Frequencies and Weightings (p. 1882)).

1/3-Octave Band (Hz)

is a range of discrete frequency bands for different 1/3 octaves within the threshold of hearing.

Table 31.3: Octave Band Frequencies and Weightings

Lower Freq. (Hz)	Center Freq. (Hz)	Upper Freq. (Hz)	dB A	dB B	dB C
11	16	22	-56.7	-28.5	-8.5
22	31.5	45	-39.4	-17.1	-3.0
45	63	90	-26.2	-9.3	-0.8
90	125	180	-16.1	-4.2	-0.2
180	250	355	-8.6	-1.3	0.0
355	500	710	-3.2	-0.3	0.0
710	1000	1400	0.0	0.0	0.0
1400	2000	2800	1.2	-0.1	-0.2
2800	4000	5600	1.0	-0.7	-0.8
5600	8000	11200	-1.1	-2.9	-3.0
11200	16000	22400	-6.6	-8.4	-8.5

Further graphical customizations for the x-axis are available by clicking **Axes...**. For more information, see Modifying Axis Attributes (p. 1852).

31.11.4.3.3. Specifying Output Options

You can write out the FFT data directly to a file by choosing the **Write FFT to File** option under **Options** in the Fourier Transform Dialog Box (p. 2602). Once the **Write FFT to File** option is selected, click **Write FFT** to display a file selection dialog box where you can choose a file and/or a location to hold the FFT data. If **Acoustics Analysis** is selected, the overall sound pressure level will be calculated in dB (based on the **Reference Acoustic Pressure**) and displayed in the console at this time.

Further customizations for how the FFT data is displayed are available by clicking **Curves...**. For more information, see Modifying Curve Attributes (p. 1854).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.11.4.3.4. Specifying a Windowing Technique

You can use the various windowing techniques described in Windowing (p. 1873) by selecting any of the **Window** options in the Plot/Modify Input Signal Dialog Box (p. 2605). By default, **None** is selected so that no windowing technique is applied.

31.11.4.3.5. Specifying Labels and Titles

You can assign a title for your FFT plot using the **Plot Title** text field. You can also assign y-axis and x-axis labels for your FFT plot using the **Y Axis Label** and **X Axis Label** text fields, respectively. By default,ANSYS Fluent assigns the **Y Axis Label** and the **X Axis Label** to the particular selection of **Y Axis Function** and **X Axis Function**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 32: Reporting Alphanumeric Data

ANSYS Fluent provides tools for computing and reporting integral quantities at surfaces and boundaries. These tools enable you to find the mass flow rate and heat transfer rate through boundaries, the forces and moments on boundaries, and the area, integral, flow rate, average, and mass average (among other quantities) on a surface or in a volume. In addition, you can print histograms of geometric and solution data, set reference values for the calculation of non-dimensional coefficients, and compute projected surface areas. You can also print or save a summary report of the models, boundary conditions, and solver settings in the current case. These features are described in the following sections.

32.1. Reporting Conventions
32.2. Monitoring and Reporting Solution Data
32.3. Creating Output Parameters
32.4. Fluxes Through Boundaries
32.5. Forces on Boundaries
32.6. Projected Surface Area Calculations
32.7. Surface Integration
32.8. Volume Integration
32.9. Histogram Reports
32.10. Discrete Phase
32.11. S2S Information
32.12. Reference Values
32.13. Summary Reports of Case Settings
32.14. System Resource Usage

Reporting tools for the discrete phase are described in Postprocessing for the Discrete Phase (p. 1329).

32.1. Reporting Conventions

For 2D problems, ANSYS Fluent computes all integral quantities for a unit depth equivalent to 1 meter. This value can be adjusted to match the specific dimension of your application only by manually revising the **Depth** in the Reference Values Task Page (p. 2502) (see Reference Values (p. 1924)).

Important

The default value of **Depth** will be equivalent to 1 meter, even if the units are changed for **depth** in the Set Units Dialog Box (p. 2166) (for example, if the units for **depth** are changed to cm in the Set Units Dialog Box (p. 2166), the value of **Depth** in the Reference Values Task Page (p. 2502) will be 100 cm).

For axisymmetric problems, all integral quantities are computed for an angle of 2π radians.

32.2. Monitoring and Reporting Solution Data

During the course of a simulation, you may want to monitor, print, write, and/or plot solution data; this is done using report definitions. A report definition is an object that specifies a certain quantity or set

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of values to be computed at the end of a solver timestep or iteration. You can then choose to have multiple report definitions printed to the console, written to a single file, or plotted in the same window.

You can use report definitions to monitor convergence, verify mesh independence, and record time-histories. These report definitions can monitor the value of field variables or functions through specified operations, such as surface, volume, force, moment, and flux integrals.

Once you create report definitions, they can be added to a report file. Report files can include multiple report definitions of varied types. Multiple report definitions can be plotted in the same graphics window by adding them to a single report plot. One report definition can be included in multiple report plots and files. For additional information on report files and plots, see Report Files and Report Plots (p. 1903).

32.2.1. Creating Report Definitions

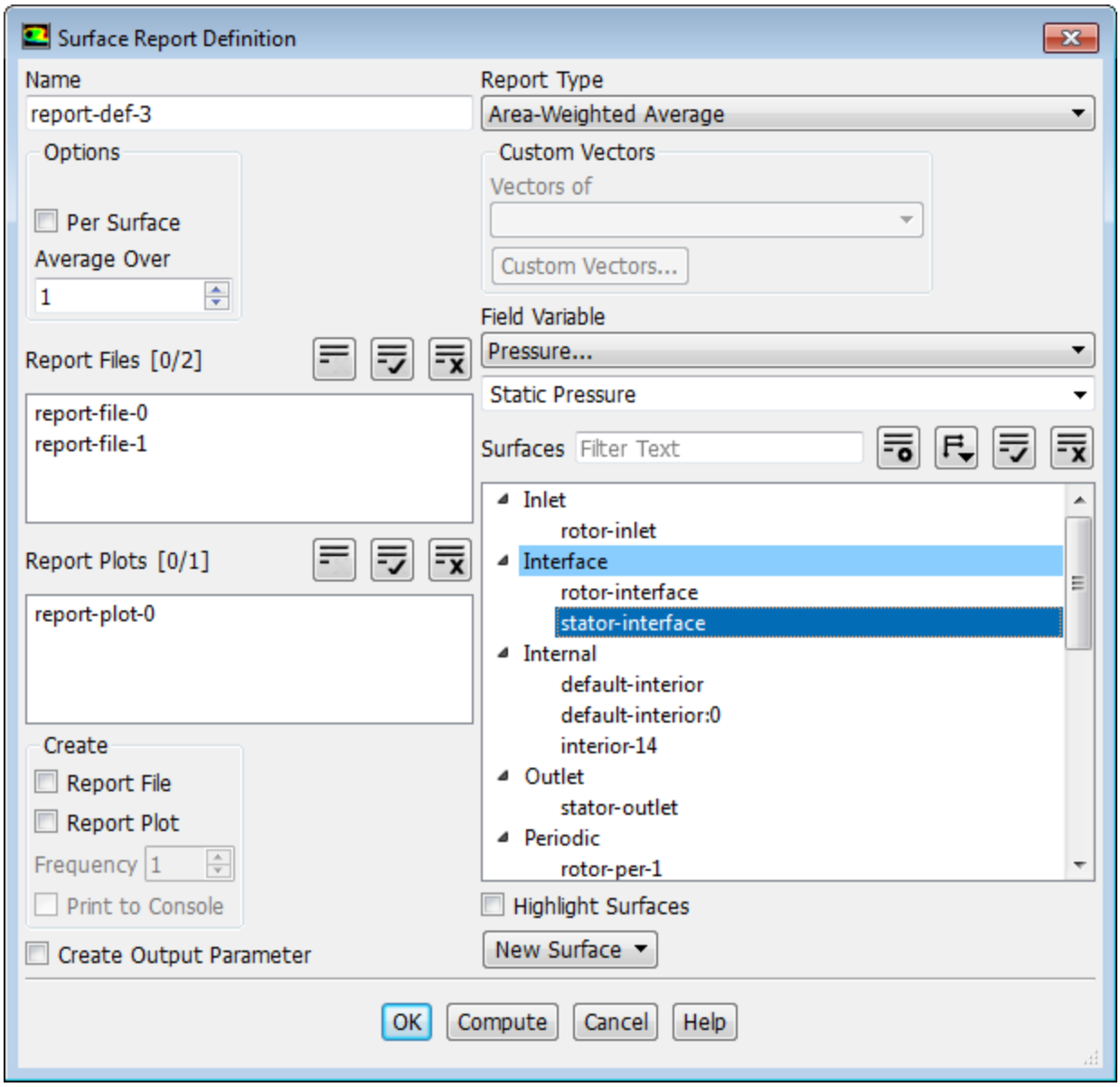
You can create report definitions from multiple locations in Fluent, allowing you to quickly define report definitions, regardless of where you are in your workflow. The primary method to create a report definition is through the ribbon. For example:

Solving → **Reports** → **Definitions** → **New** → **Surface Report** → **Area-Weighted Average...**

Opens the **Surface Report Definition** dialog box.

Alternative methods for accessing this dialog box are through the tree, by right-clicking the ***Report Definition*** *branch, or through the* ***New*** *drop-down list in the* *Figure 32.1: Report Definitions Dialog Box (p. 1889)*.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Many of the setup steps are the same or similar when setting up the various types of report definitions.

To setup the Surface Report Definition dialog box:

1. Enter a name for the report definition under **Name**.
2. If you want the value reported on individual surfaces rather than the net results from a group of surfaces, enable the **Per Surface** option.
3. (optional) To have Fluent calculate a running average for the **Surface Report Definition** you can enter a positive integer greater than 1 (the default) for **Average Over**.

   Specifying a number greater than 1 means that Fluent will print, plot, and write the running average value of the selected variable instead of the current value of the same variable.

   The value reported is averaged over the last N *iterations/time steps, where* N *is your specified* ***Average Over*** *value. When the iteration/time step number is lower than* N*, Fluent calculates the average of the available variable values.*

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. If you want the report definition written to a file or plotted, either select an existing report file or plot from the **Report Files** or **Report Plots** lists, or create a new report file or plot by enabling **Report File** and/or **Report Plot** in the **Create** group box.

5. If you want the report definition available as an output parameter, enable **Create Output Parameter**.

6. Choose the surface report definition integration method by selecting **Area**, **Integral**, **Standard Deviation**, **Flow Rate**, **Mass Flow Rate**, **Volume Flow Rate**, **Area-Weighted Average**, **Mass-Weighted Average**, **Sum**, **Uniformity Index - Mass Weighted**, **Uniformity Index - Area Weighted**, **Facet Average**, **Facet Minimum**, **Facet Maximum**, **Vertex Average**, **Vertex Minimum**, **Vertex Maximum**, **Custom Vector Based Flux**, **Custom Vector Flux**, or **Custom Vector Weighted Average** from the **Report Type** drop-down list. These methods are described in Surface Integration (p. 1920).

7. If you selected **Custom Vector Based Flux**, **Custom Vector Flux**, or **Custom Vector Weighted Average** from the **Report Type** drop-down list, you must define the vectors in the **Custom Vectors** group box.

8. Unless you are reporting a mass flow rate or volume flow rate, specify the variable or function to be integrated in the **Field Variable** drop-down list. First select the desired category in the upper drop-down list. You can then select one of the related quantities in the lower list. (See Field Function Definitions (p. 1931) for an explanation of the variables in the list.) If you are running a multiphase simulation you may need to select the phase of interest (or **mixture**) from the **Phase** drop-down list (depending on the field variable selected).

   Note

   - In general, you can select any arbitrary field variable for each report type. However, not all possible combinations are physically meaningful.
   - For multiphase flow, you should use only the **Mass Flow Rate** integral to compute the mass flow rate of the mixture or for an individual phase. Do not use the **Flow Rate** integral of the **Volume Fraction** field variable as this will result in erroneous results.

9. Select the **Surfaces** where you want Fluent to evaluate the specified variable.

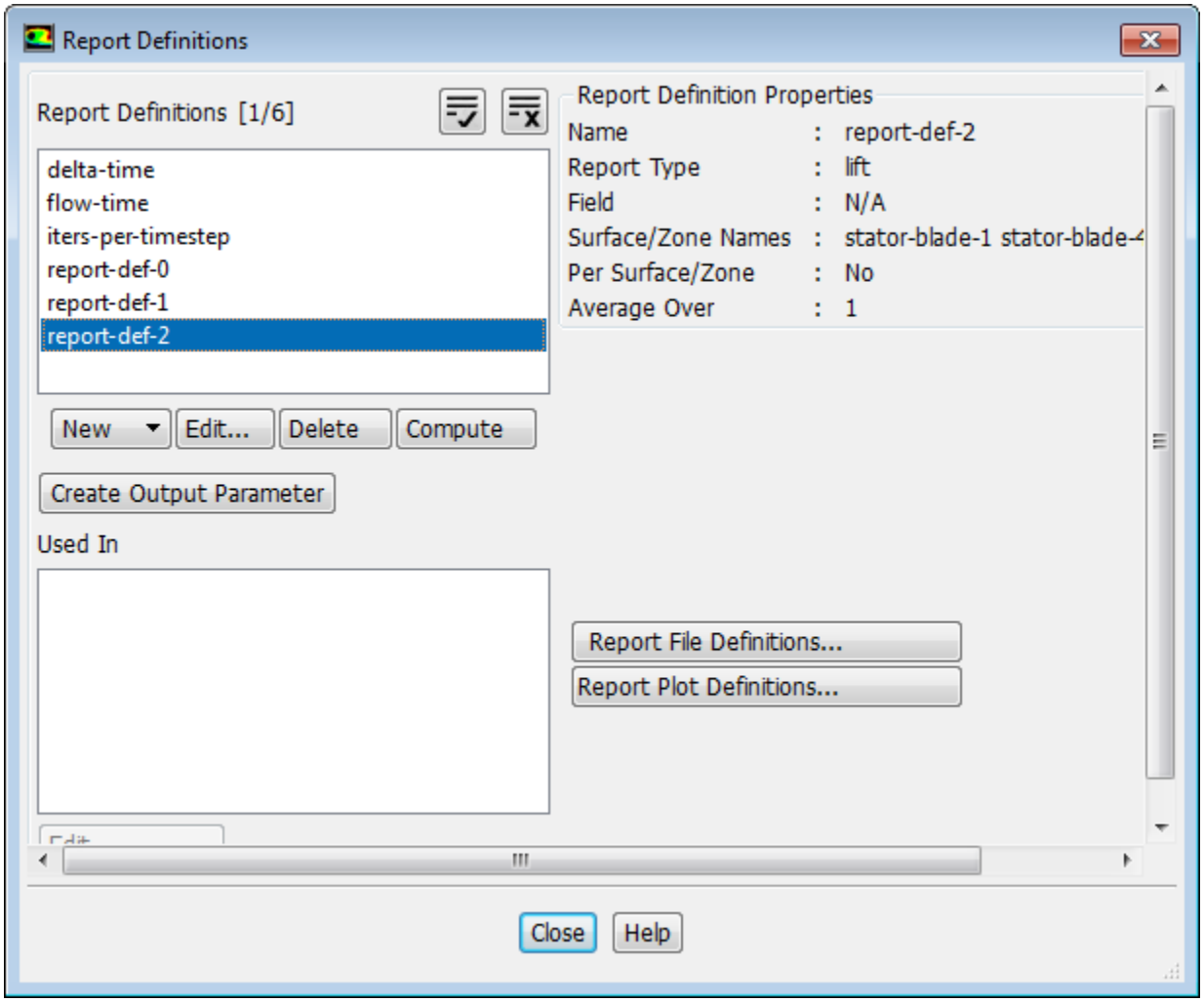
10. You have the option to highlight a selected surface and have it displayed in the graphics window by enabling **Highlight Surfaces**.

11. Click **OK** to create the report definition and close the dialog box.

Once you have created a report definition, it appears in the Figure 32.1: Report Definitions Dialog Box (p. 1889).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 32.1: Report Definitions Dialog Box



You can use this dialog box to review and modify the properties of the existing report definitions.

The following sections detail setup instructions that are specific to each type of report definition.

32.2.1.1. Surface Report definitions
32.2.1.2. Volume Report definitions
32.2.1.3. Force and Moment Report definitions
32.2.1.4. Flux Report Definition
32.2.1.5. DPM Report Definition
32.2.1.6. User Defined Report Definition
32.2.1.7. Expression Report Definition

32.2.1.1. Surface Report definitions

You can use the Figure 32.2: Surface Report Definition Dialog Box (p. 1891) to create surface report definitions. There are many different types of surface-based report definitions that you can create, including:

- Integral
- Standard Deviation

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

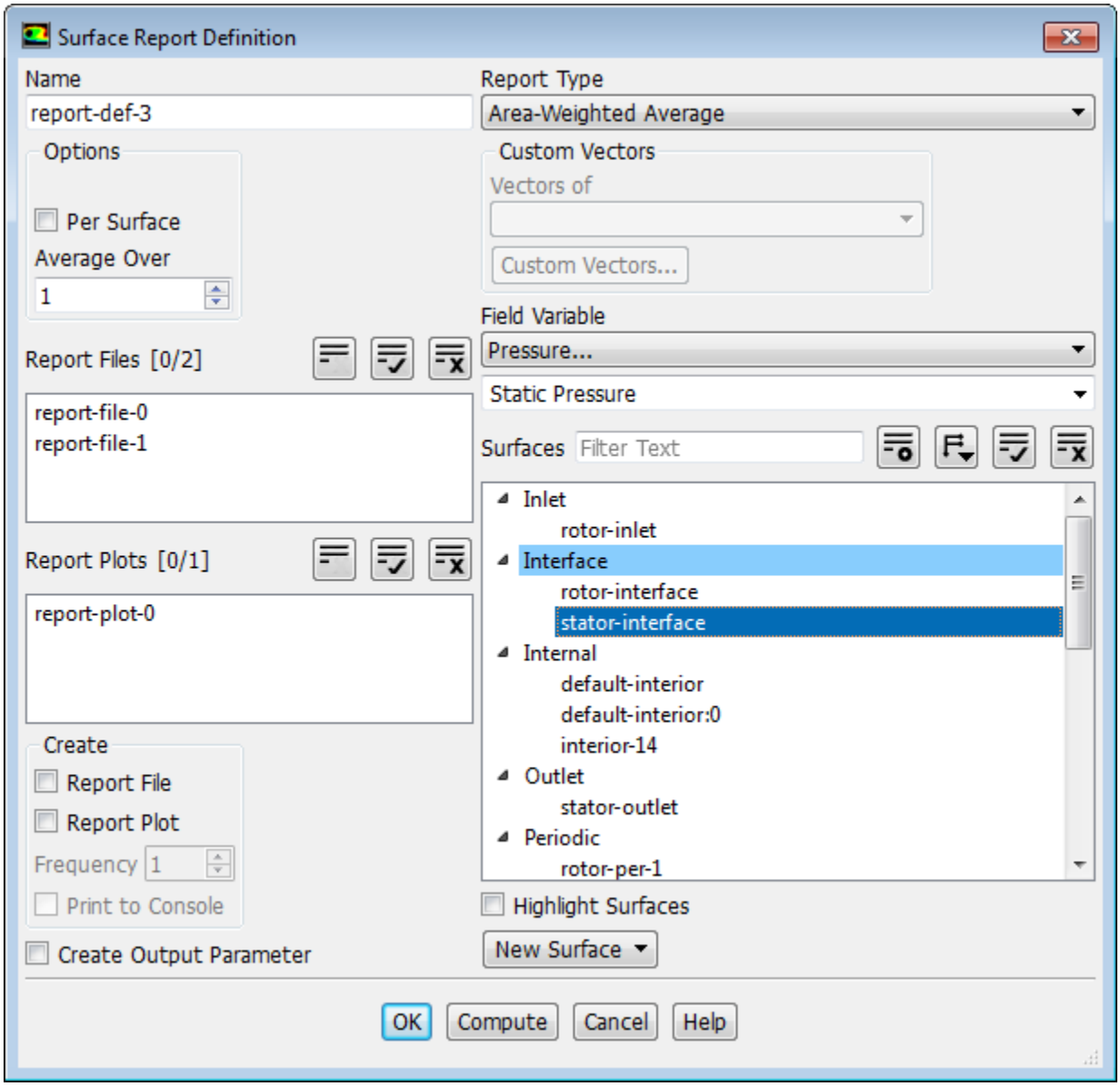
- Flow Rate
- Mass Flow Rate (non-field based)
- Volume Flow Rate (non-field based)
- Area-Weighted Average
- Mass-Weighted Average
- Sum
- Uniformity Index — Mass Weighted
- Uniformity Index — Area Weighted
- Facet Average
- Facet Minimum
- Facet Maximum
- Custom Vector Based Flux (custom-vector based)
- Custom Vector Flux (custom-vector based, non-field based)
- Custom Vector Weighted Average (custom-vector based)

To plot or write the surface report definition data you must use a **Report File** or **Report Plot**. If a surface report definition is already included in a report file/plot, then that report file/plot will be highlighted. If you de-select a report file/plot that had included the current report definition and click **OK**, the report definition will be removed from that report file/plot. For additional information on report files and plots, see Report Files and Report Plots (p. 1903).

Solving → **Reports** → **Definitions** → **New** → **Surface Report**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 32.2: Surface Report Definition Dialog Box



The procedure for creating surface report definitions is covered in Creating Report Definitions (p. 1886).

32.2.1.2. Volume Report definitions

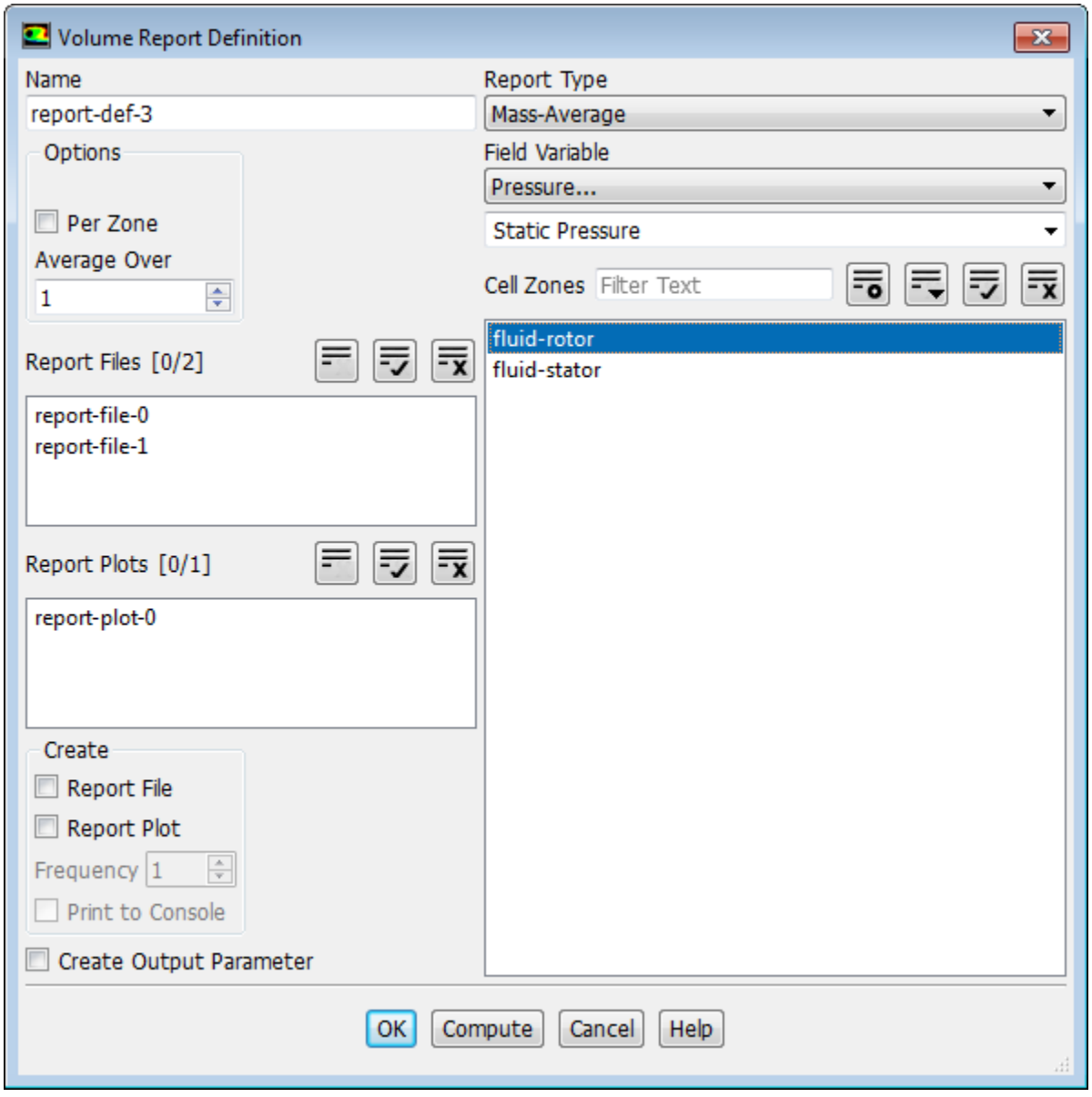
You can use the Figure 32.3: Volume Report Definition Dialog Box (p. 1892) to create volume report definitions. There are many different types of volume reports that you can create, including:

- Volume (non-field based)
- Sum
- Sum*2π (only axi-symmetric)
- Max
- Min

- Volume Integral
- Volume Average
- Mass Integral
- Mass Average
- Mass (non-field based)

Solving → Reports → Definitions → New → Volume Report

Figure 32.3: Volume Report Definition Dialog Box



The common steps for creating a volume report definition are covered in Creating Report Definitions (p. 1886), and the volume report definition-specific steps are as follows:

1. If you want the value reported on individual wall zones rather than the net results from a group of wall zones, enable the **Per Zone** option.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Choose the integration method for the volume report definition by selecting **Volume**, **Sum**, **Sum*2Pi** (2D axisymmetric cases only), **Max**, **Min**, **Volume Integral**, **Volume-Average**, **Mass Integral**, **Mass-Average**, or **Mass** in the **Report Type** drop-down list. These methods are described in Volume Integration (p. 1922).

3. Specify the variable or function to be integrated in the **Field Variable** drop-down list. First select the desired category in the upper drop-down list. You can then select one of the related quantities in the lower list. (See Field Function Definitions (p. 1931) for an explanation of the variables in the list.)

4. Select the **Cell Zones** where you want Fluent to evaluate the specified variable.

5. Click **OK** to create the report definition and close the dialog box.

32.2.1.3. Force and Moment Report definitions

Monitoring force coefficients can be useful when you are calculating external aerodynamics, for example, and are especially interested in the lift coefficient. By monitoring these values you may also be able to stop the calculation early and reduce the processing time, as sometimes the force and moment coefficients converge before the residuals have decreased three orders of magnitude. (In such an instance, you should be sure to check the mass flow rate and heat transfer rate as well, to ensure that the mass and energy are being suitably conserved. This is accomplished using either the Figure 32.4: Drag Report Definition Dialog Box (p. 1894), the Figure 32.5: Lift Report Definition Dialog Box (p. 1895), or the Figure 32.6: Moment Report Definition Dialog Box (p. 1896)).

To begin setting up force or moment report definitions, first enter appropriate values in the **Reference Values** task page, as described in Reference Values (p. 1924). The relevant values include the following:

- The force coefficients use the reference area, density, and velocity.
- The moment coefficients use the reference area, density, velocity and length.

For information about how these coefficients are calculated, see Computing Forces, Moments, and the Center of Pressure in the Fluent Theory Guide.

Tip

You can have Fluent report unscaled force values by selecting **Drag Force** or **Lift Force** in the **Report Output Type** group box of the **Drag Report Definition** or **Lift Report Definition** dialog boxes, respectively.

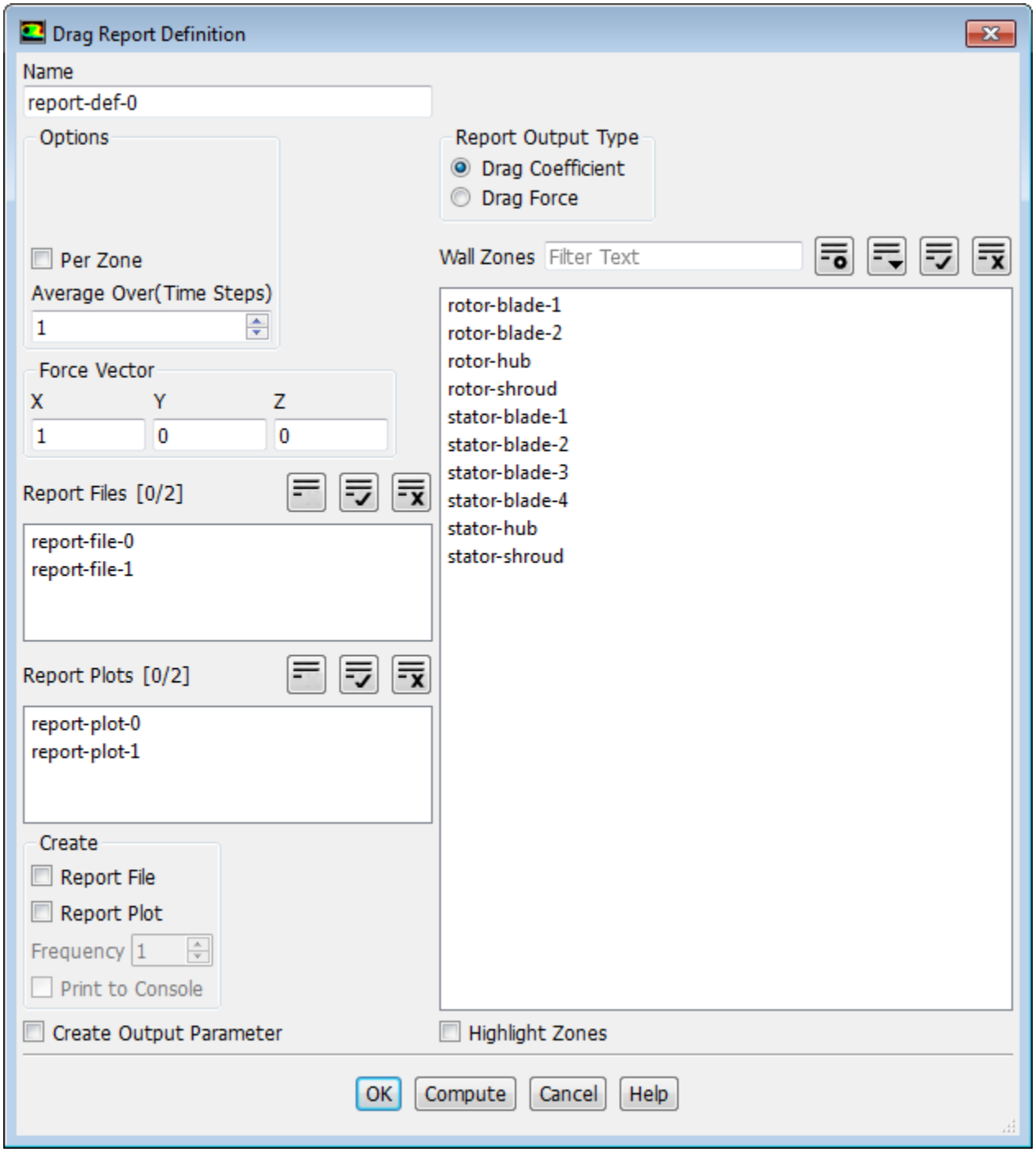
Next, open the appropriate dialog box through the **Solving** tab of the ribbon.

Solving → **Reports** → **Definitions** → **New** → **Force Report**

Select either **Drag...**, **Lift...**, or **Moment...** to open the Figure 32.4: Drag Report Definition Dialog Box (p. 1894), Figure 32.5: Lift Report Definition Dialog Box (p. 1895), and Figure 32.6: Moment Report Definition Dialog Box (p. 1896). Note that you can only access one of these report definition dialog boxes at a time, though multiple report definitions of the same type can be used during the simulation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 32.4: Drag Report Definition Dialog Box



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 32.5: Lift Report Definition Dialog Box

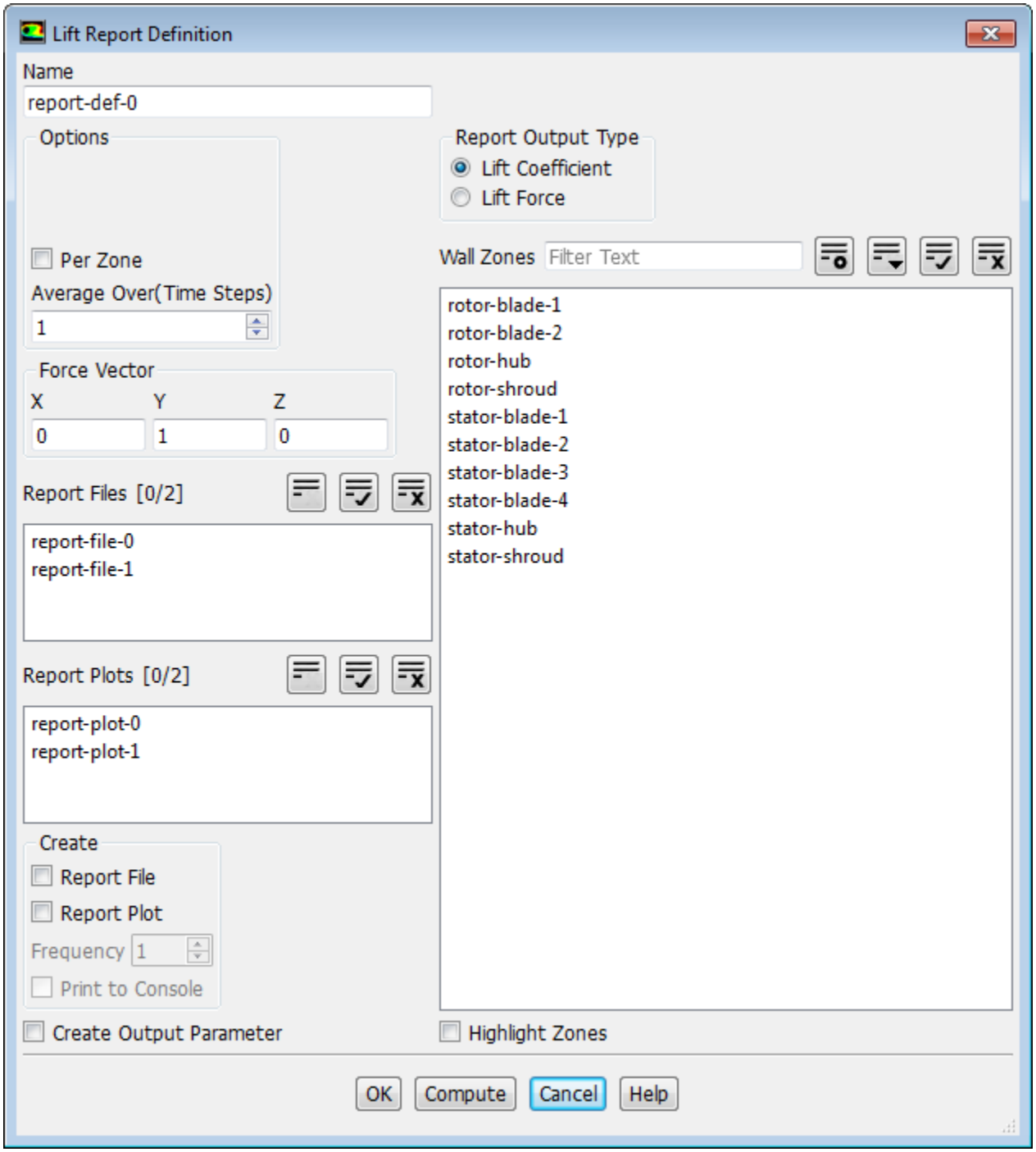
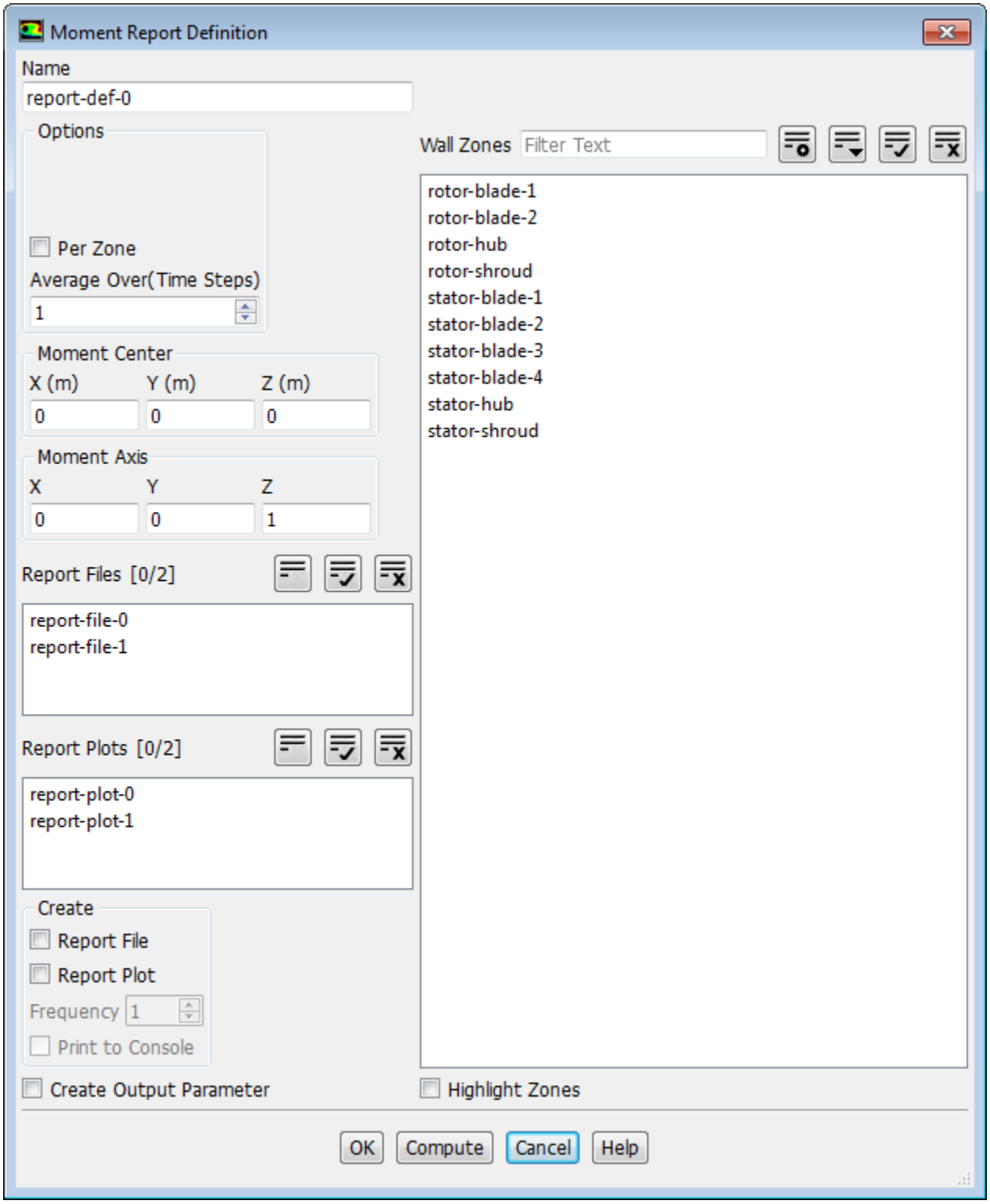


Figure 32.6: Moment Report Definition Dialog Box



The common steps for creating a force or moment report definition are covered in Creating Report Definitions (p. 1886), and the force/moment report definition-specific steps are as follows:

1. If you want to report the force or moment coefficient data from individual wall zones rather than the net results from a group of wall zones, enable the **Per Zone** option.

2. Depending on the coefficient that will be monitored, perform one of the following steps:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- In the **Drag Report Definition** or **Lift Report Definition** dialog box, enter the **X**, **Y**, and **Z** components of the **Force Vector** along which the forces will be computed. By default, the **Force Vector** for the drag coefficient is a unit vector in the x direction, whereas for the lift coefficient it is a unit vector in the y direction.

- In the **Moment Report Definition** dialog box, enter the Cartesian coordinates (**X**, **Y**, and **Z**) of the **Moment Center**, about which moments will be computed. The default **Moment Center** is (**0**, **0**, **0**). You also need to enter the **X**, **Y**, and **Z** components for the **Moment Axis**, along which the moment coefficient will be calculated. By default, the **Moment Axis** is defined as a unit vector in the z direction.

3. Select the appropriate wall zone(s) for computation of the coefficient(s).

4. You have the option to highlight a selected boundary zone and have it displayed in the graphics window by enabling **Highlight Zone**.

5. Click **OK** to save the report definition settings.

6. If you need to revise the settings of a particular report definition after they have been saved, double-click the report definition in the tree.

32.2.1.4. Flux Report Definition

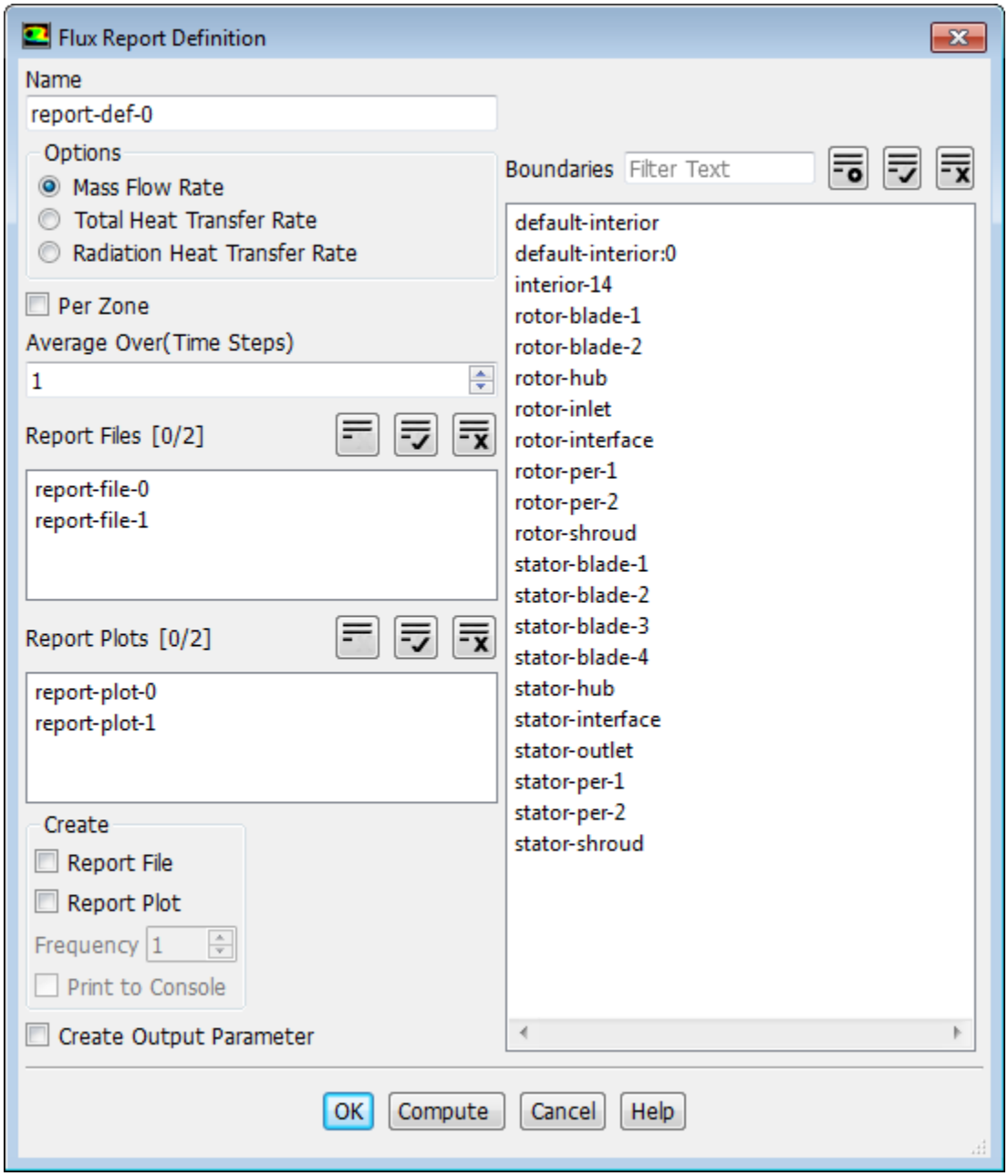
You can use the Figure 32.7: Flux Report Definition Dialog Box (p. 1898) to create flux report definitions. There are many different types of flux-based report definitions that you can create, including:

- Mass Flow Rate
- Total Heat Transfer Rate
- Total Sensible Heat Transfer Rate
- Radiation Heat Transfer Rate
- Film Mass Flow Rate
- Film Heat Transfer Rate
- DPM Mass Source
- DPM Enthalpy Source
- DPM Sensible Enthalpy Source

Solving → **Reports** → **Definitions** → **New** → **Flux Report**

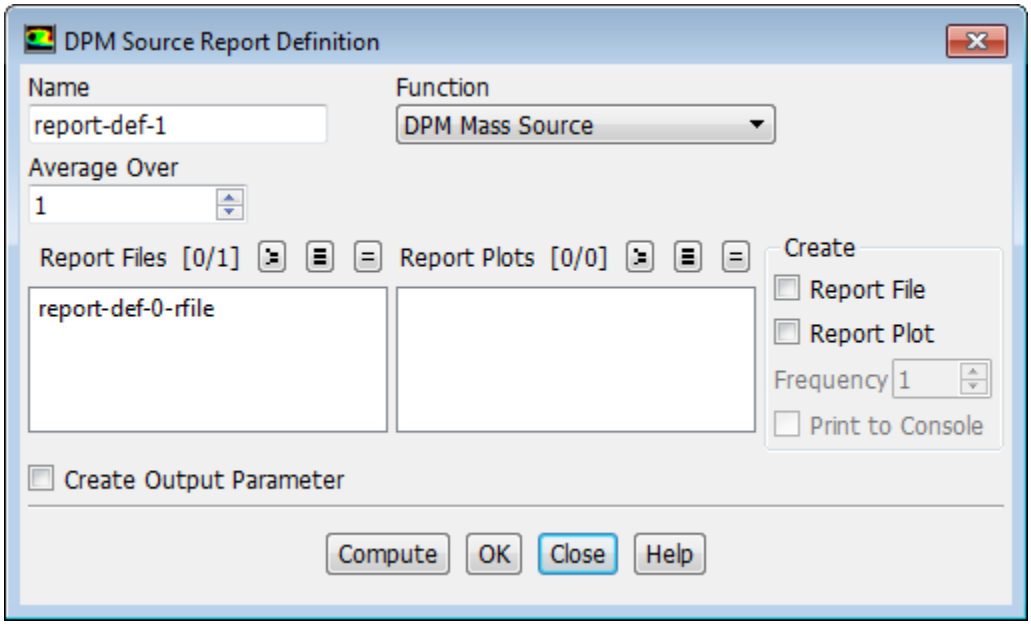
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 32.7: Flux Report Definition Dialog Box



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 32.8: DPM Source Report Definition Dialog Box



The common steps for creating a flux report definition are covered in Creating Report Definitions (p. 1886), and the flux report definition-specific steps are as follows:

1. Select or confirm the type of flux report.
2. Select the appropriate boundaries for the flux calculations.
3. Click **OK** to save the report definition settings.
4. If you need to revise the settings of a particular report definition after they have been saved, double-click the report definition in the tree.

For additional information on the available options for DPM source report definitions, see DPM Source Report Definition (p. 2690) in the reference guide.

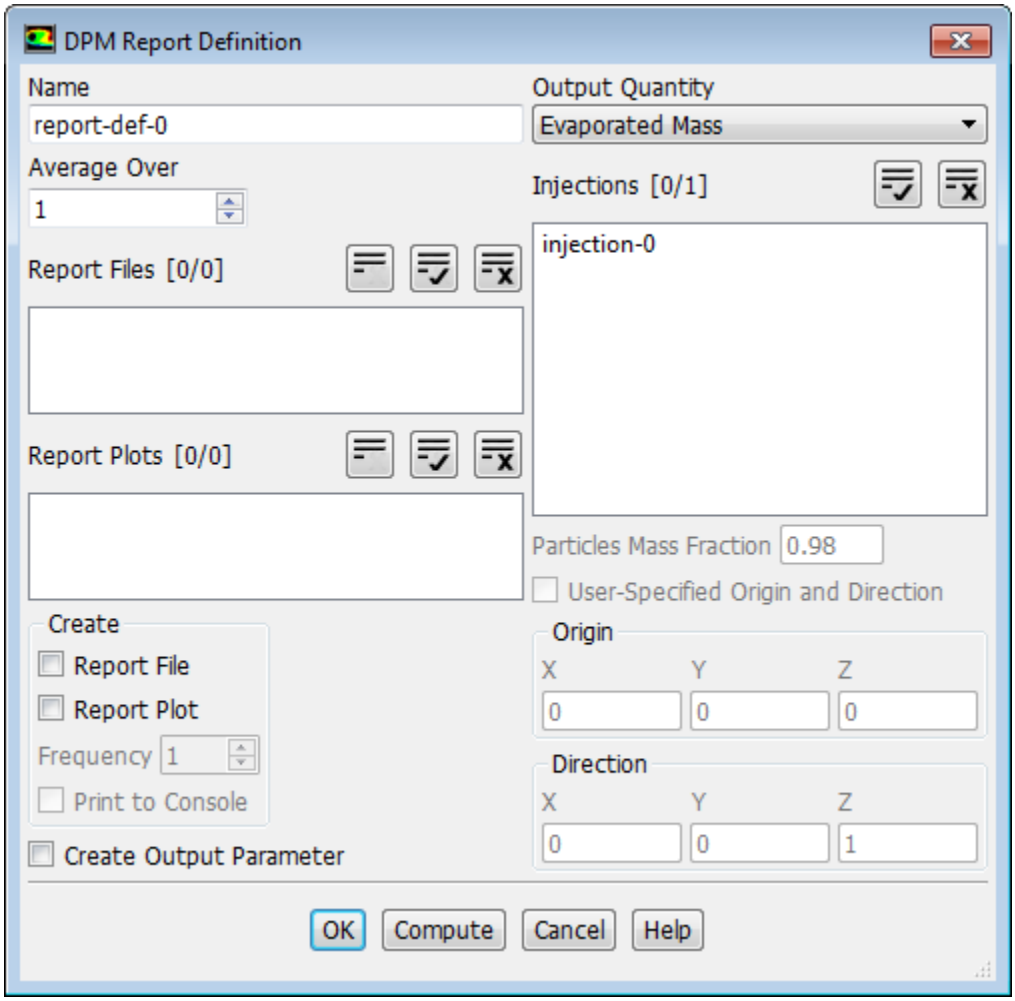
32.2.1.5. DPM Report Definition

You can use the DPM Report Definition dialog box to create DPM report definitions. The following types of DPM reports are available:

- Injected Mass (*unsteady particle tracking only*)
- Mass in Domain (*unsteady particle tracking only*)
- Escaped Mass
- Evaporated Mass
- Penetration Length (*unsteady particle tracking only*)

Solving → Reports → Definitions → New → DPM Report

Figure 32.9: DPM Report Definition Dialog Box



The common steps for creating a DPM report definition are covered in Creating Report Definitions (p. 1886), and the DPM report definition-specific steps are as follows:

1. Select or confirm the type of DPM report.

2. For the **Escaped Mass** report, if you want to have the escaped mass calculated per injection, enable the text command `report/dpm-zone-summaries-per-injection`.

   The **Injections** selection list will then appear in the **DPM Report Definition** dialog box.

3. From the **Injections** list, select the appropriate injections for the DPM report.

   Note

   For the **Escaped Mass** report, this step is relevant only if you have enabled the calculation of the escaped mass per injection.

4. For the **Escaped Mass** report, select the boundary(s) where the injected particles exit the domain from the **Boundaries** selection list.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. For the **Penetration Length** report, you can specify the **Particles Mass Fraction**. You can also enable **User-Specified Origin and Direction** to specify those coordinates.

6. For unsteady particle tracking, you can select **Show Mass Flow/Change Rate** (default) for the **Injected Mass**, **Mass in Domain**, **Escaped Mass**, and **Evaporated Mass** reports if you want to create a report for the mass flow rate rather than the total mass.

7. Click **OK** to save the report definition settings.

8. If you need to revise the settings of a particular report definition after they have been saved, double-click the report definition in the tree.

Important

For unsteady particle tracking, if you want to select one or more injection(s) for any **Escaped Mass** report, you must enable the `report/dpm-zone-summaries-per-injection` text command before the first particles are injected; otherwise, some of the reported values may become inconsistent.

For additional information on the available options, see the DPM Report Definition Dialog Box (p. 2687) in the reference guide.

32.2.1.6. User Defined Report Definition

User defined report definitions allow you to create functions that return values that can be plotted, printed, written, or used in other expressions. User defined report definitions can also be used in the **Convergence Conditions** (Convergence Conditions (p. 1635)).

Creating a user defined report definition is a two step process:

1. Create a user defined report definition function.

2. Hook the user defined report definition function to the user defined report definition.

32.2.1.6.1. User Defined Report Definition Function

User defined report definition functions are defined with the `DEFINE_REPORT_DEFINITION_FN` macro, which takes the function name as the argument. The function must return a real value. Once written, it should be compiled and loaded the same way as other user-defined functions, see Interpreting and Compiling UDFs in the *Fluent Customization Manual* for more information.

The function body should be constructed similarly to any C function that returns a real value. Any valid C code can be included in the function body.

`DEFINE_REPORT_DEFINITION_FN` can use all of the APIs available for other user-defined functions, including:

- Last evaluated values of all report definitions.

- Current values of all input parameters that have been set to be used inside user-defined functions.

See `DEFINE_REPORT_DEFINITION_FN` in the *Fluent Customization Manual* for information on writing user defined report definition functions.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.2.1.6.2. User Defined Report Definition Function Hooking

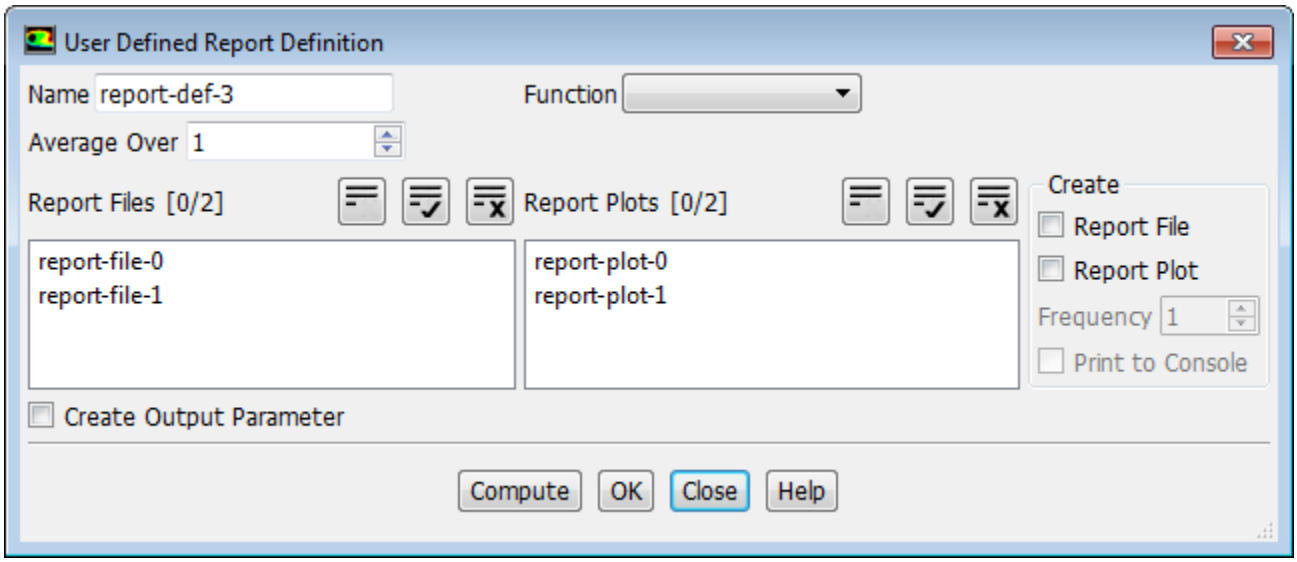
With the user defined report definition function written, you must hook this function to the user defined report definition.

The common steps for creating a user defined report definition are covered in Creating Report Definitions (p. 1886), and the user defined report definition-specific steps are as follows:

1. Open the **User Defined Report definition** dialog box.

   Solution → Report Definitions → New → User Defined

   Figure 32.10: User Defined Report Definition Dialog Box



2. Select your function from the **Function** drop-down list.

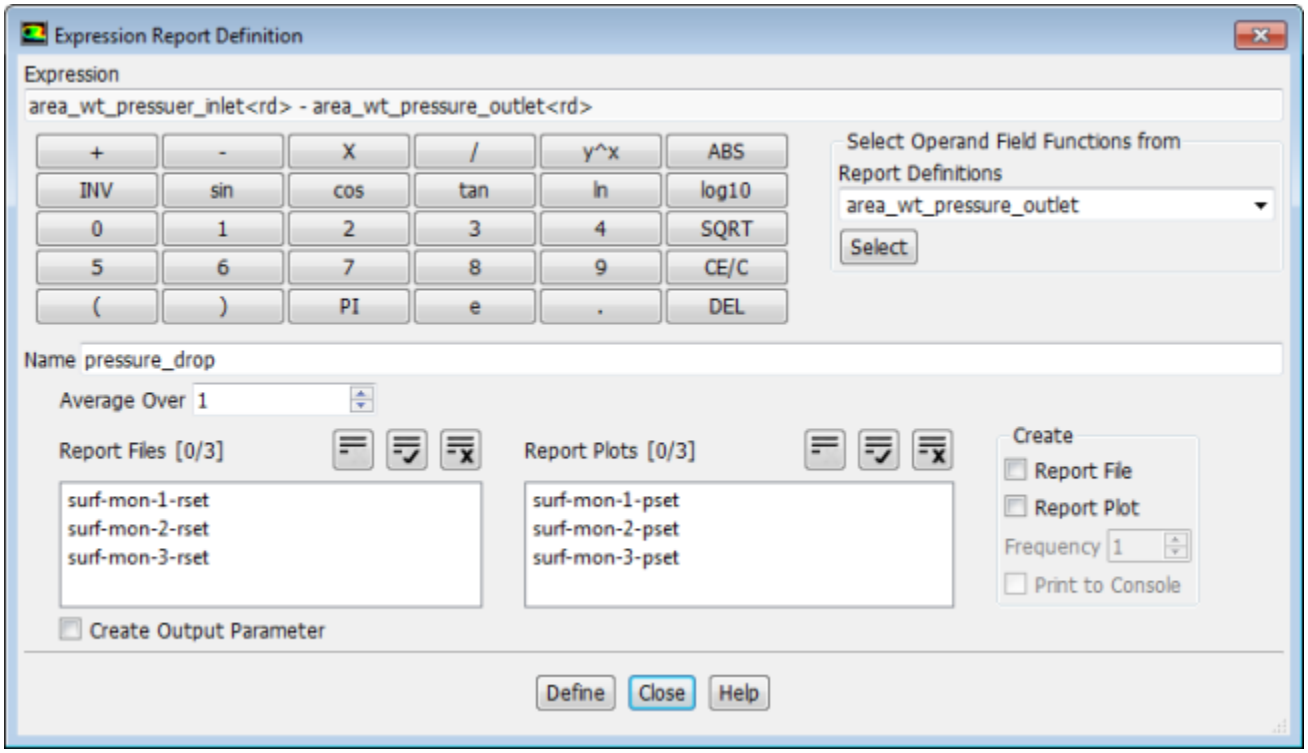
32.2.1.7. Expression Report Definition

You can create expressions from report definitions, allowing you to combine report definitions with mathematical operators to create a new report definition that can be written or plotted. For example, you can create a pressure-drop report for the difference in area-weighted pressures between an inlet and an outlet.

Solving → Reports → Definitions → New → Expression

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 32.11: Expression Report Definition Dialog Box



The common steps for creating an expression report definition are covered in Creating Report Definitions (p. 1886), and the expression report definition-specific steps are as follows:

1. Create the **Report Definition**(s) that you will use to create an **Expression**.
2. Use the calculator buttons to add functions and operators to the expression. Select report definitions from the **Report Definitions** drop-down list and click **Select** to add them to the expression formula.
3. Click **Define** to create the expression report definition.

Note

Expression report definitions cannot be copied.

32.2.2. Report Files and Report Plots

Report Files and **Report Plots** are collections of report definitions that are written to a single file (report file) or plotted to a graphics window (report plot) at specified intervals. In steady-state cases, report files are evaluated at the specified iteration, whereas in transient cases you have the option to specify the evaluation frequency based on iterations or timesteps.

*Report definitions in the same **Report Plot** must have the same units.*

Report Files Format:

```
"<report-file-name>"
"Iteration"|"Timestep" "<first-report-def-name> etc.."
("Iteration"|"Timestep" "<report-def-1-name>" "<report-def-2-name>"...)
<Iteration/Timestep> <report-def-1-value> <report-def-2-value>...
```

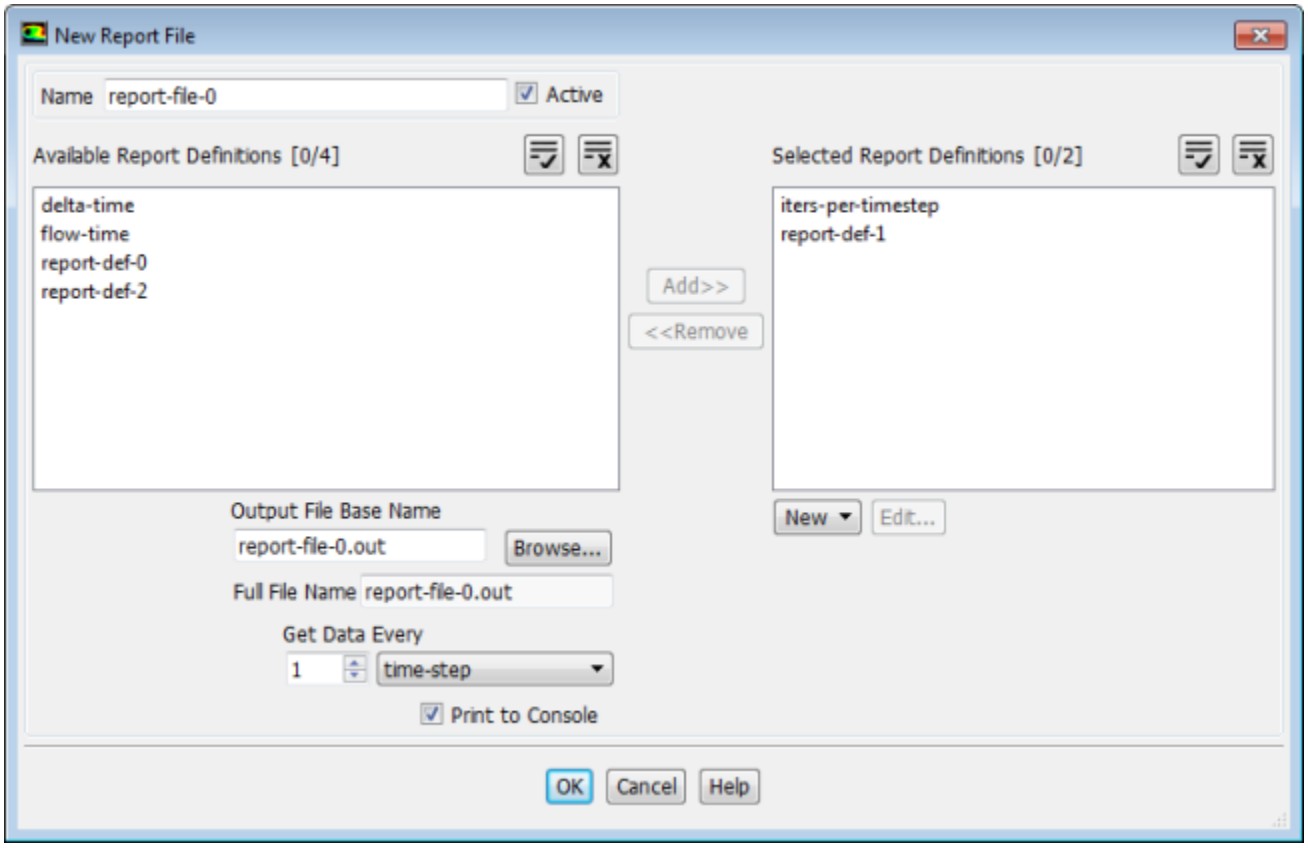
Report definitions included in a report plot are plotted in the specified window at the specified frequency. It is possible to plot the same report in multiple windows. Both report plots and report files provide the option to print to the console. You can deactivate the writing and plotting of individual report files and plots for a particular calculation; deactivated report files and plots can be re-activated at a later time.

32.2.2.1. Creating Report Files

In order to create a report file you must have first created a report definition.

Solution → Monitors → Report Files → New...

Figure 32.12: New Report File Dialog Box



1. Provide a name for how you want this report file to appear in the Fluent tree.

   *You have the option to deactivate the writing of this report file by de-selecting the **Active** check box to the right of the **Name**.*

2. Select one or more report definitions from **Available Report Defnitions** and click **Add>>**, to specify the reports you want written to the same file.

3. Provide an appropriate name in the **Output File Base Name** text box. The **Full File Name** that appears below the **Output File Base Name** field is created with a run index and iteration/time step index, appended to the base name.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Specify the frequency at which you want the report definitions written to the file.

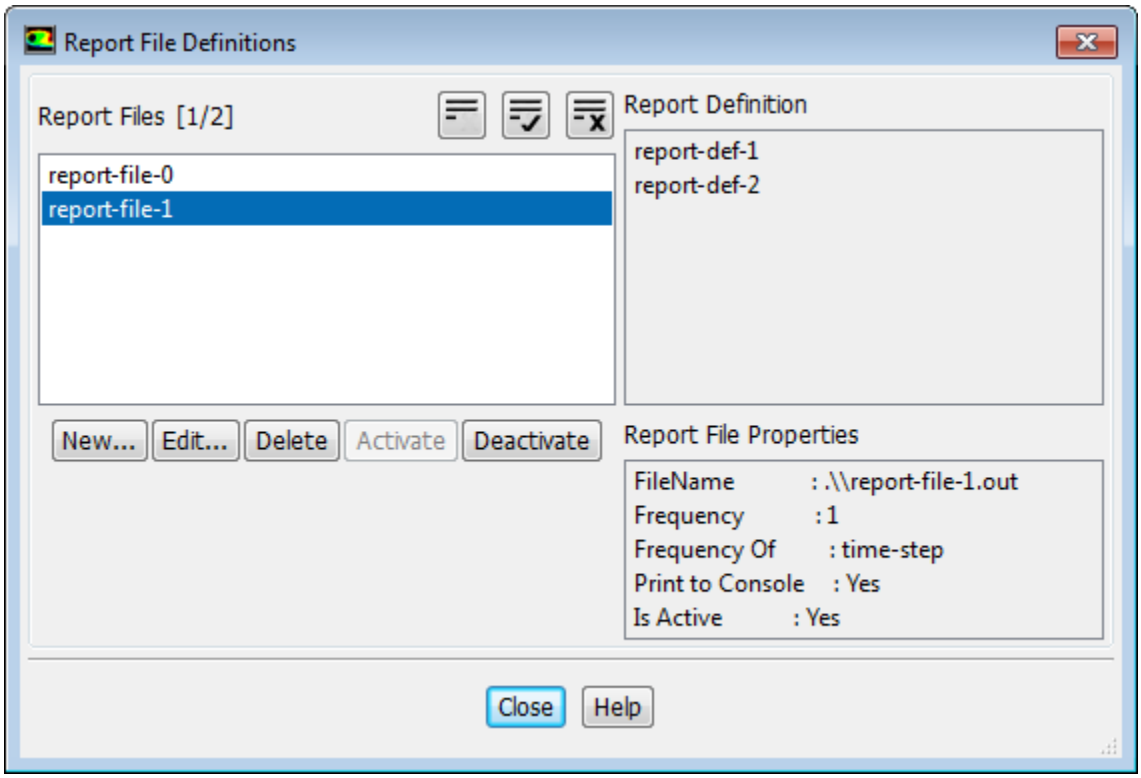
> **Note**
>
> In a transient simulation, If you specify an **Average Over** value in a report definition, you must set **Get Data Every** to **time-step**. If you set **Get Data Every** to **iteration**, Fluent will use the instantaneous value of the report definition.

5. Click **OK** to save the new report file.

You have the option to view—and edit —properties of report files that you created previously by selecting **Edit...** when you right-click **Report Files** in the tree, which opens the Figure 32.13: Report File Definitions Dialog Box (p. 1905).

Note that you can also right-click an individual report definition to modify it directly.

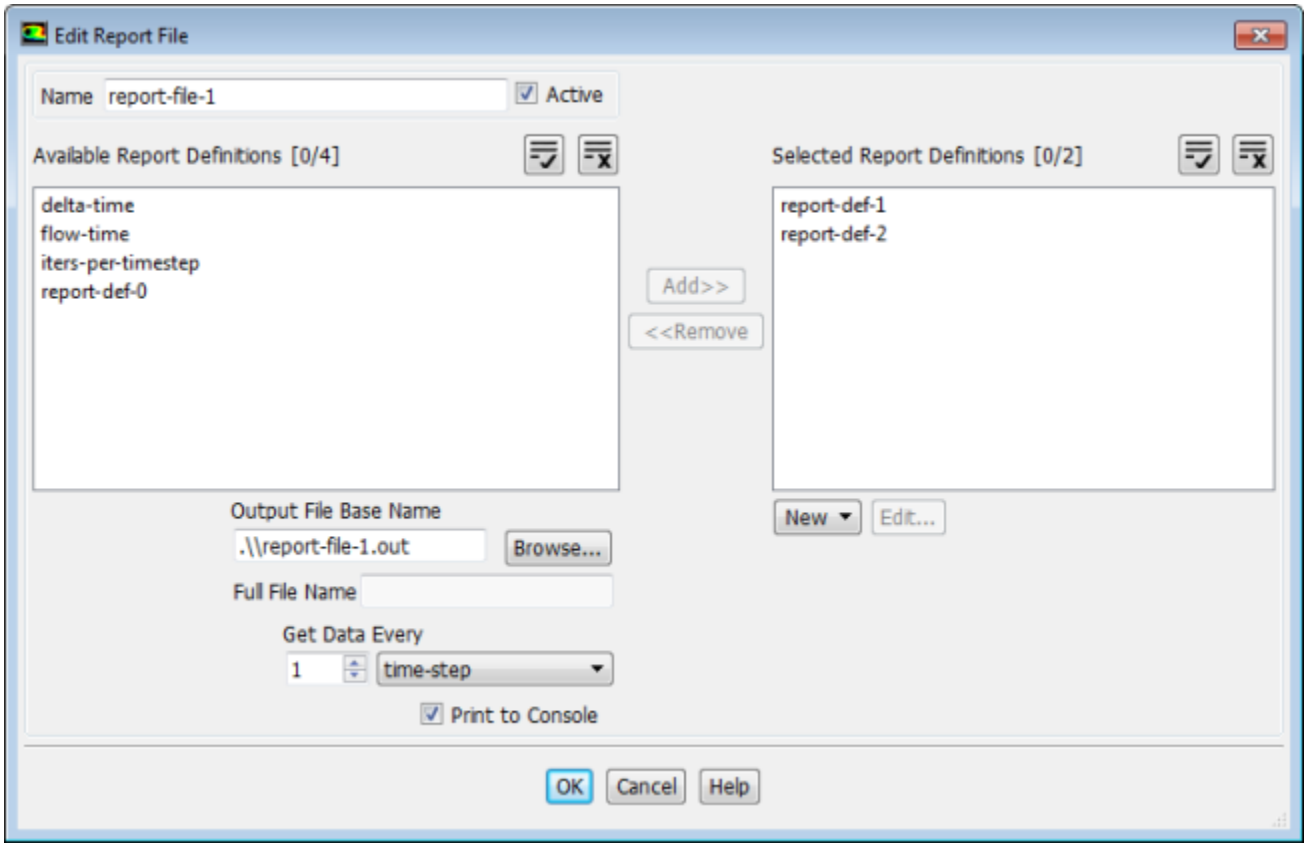
Figure 32.13: Report File Definitions Dialog Box



Selecting a report file to modify opens the Figure 32.14: Edit Report File Dialog Box (p. 1906).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 32.14: Edit Report File Dialog Box



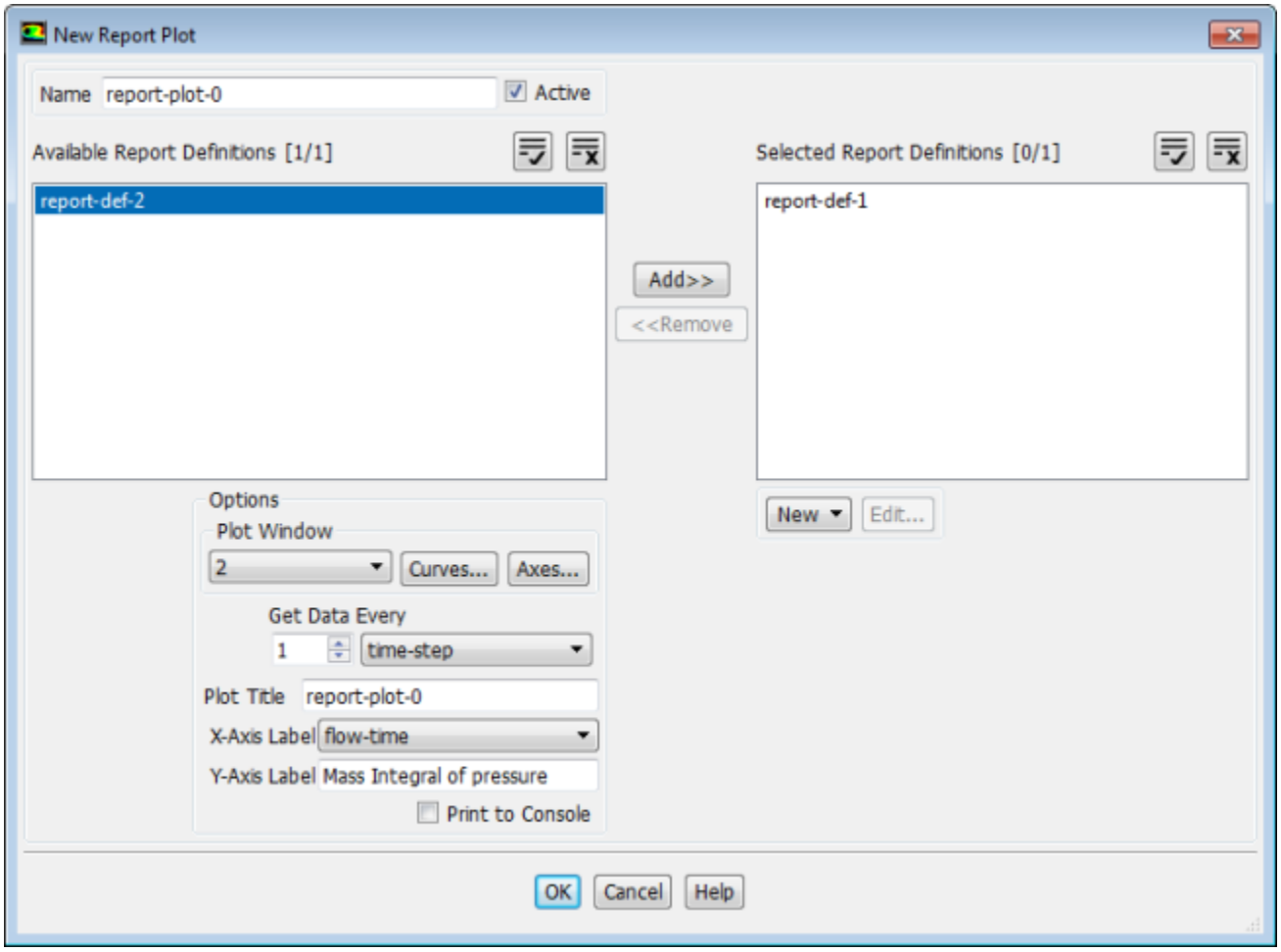
32.2.2.2. Creating Report Plots

In order to create a report plot you must have first created a report definition.

Solution → Monitors → Report Plots → New...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 32.15: New Report Plot Dialog Box



1. Provide a name for how you want this report plot to appear in the Fluent tree.

   *You have the option to deactivate the plotting of this report plot by de-selecting the **Active** check box to the right of the **Name**.*

2. Select one or more report definitions from **Available Report Defnitions** and click **Add>>**, to specify the reports you want plotted in the same window.

3. Provide an appropriate names in the **Plot Title** and **Y-Axis Label** group boxes.

4. Specify the frequency at which you want the report definitions plotted to the specified window.

   Note

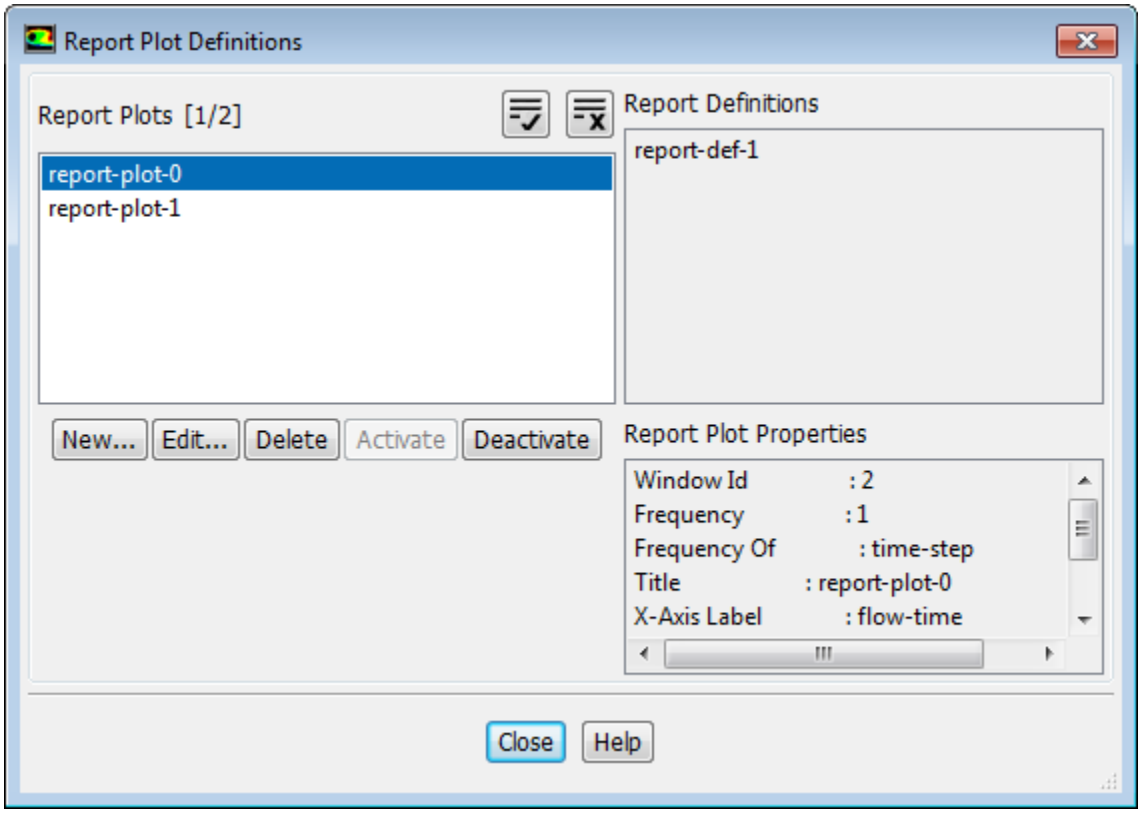
   In a transient simulation, If you specify an **Average Over** value in a report definition, you must set **Get Data Every** to **time-step**. If you set **Get Data Every** to **iteration**, Fluent will use the instantaneous value of the report definition.

5. Click **OK** to save the new report plot.

You have the option to edit report plots that you created previously by selecting **Edit...** when you right-click **Report Plots** in the tree, which opens the Figure 32.16: Report Plot Definitions Dialog Box (p. 1908).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

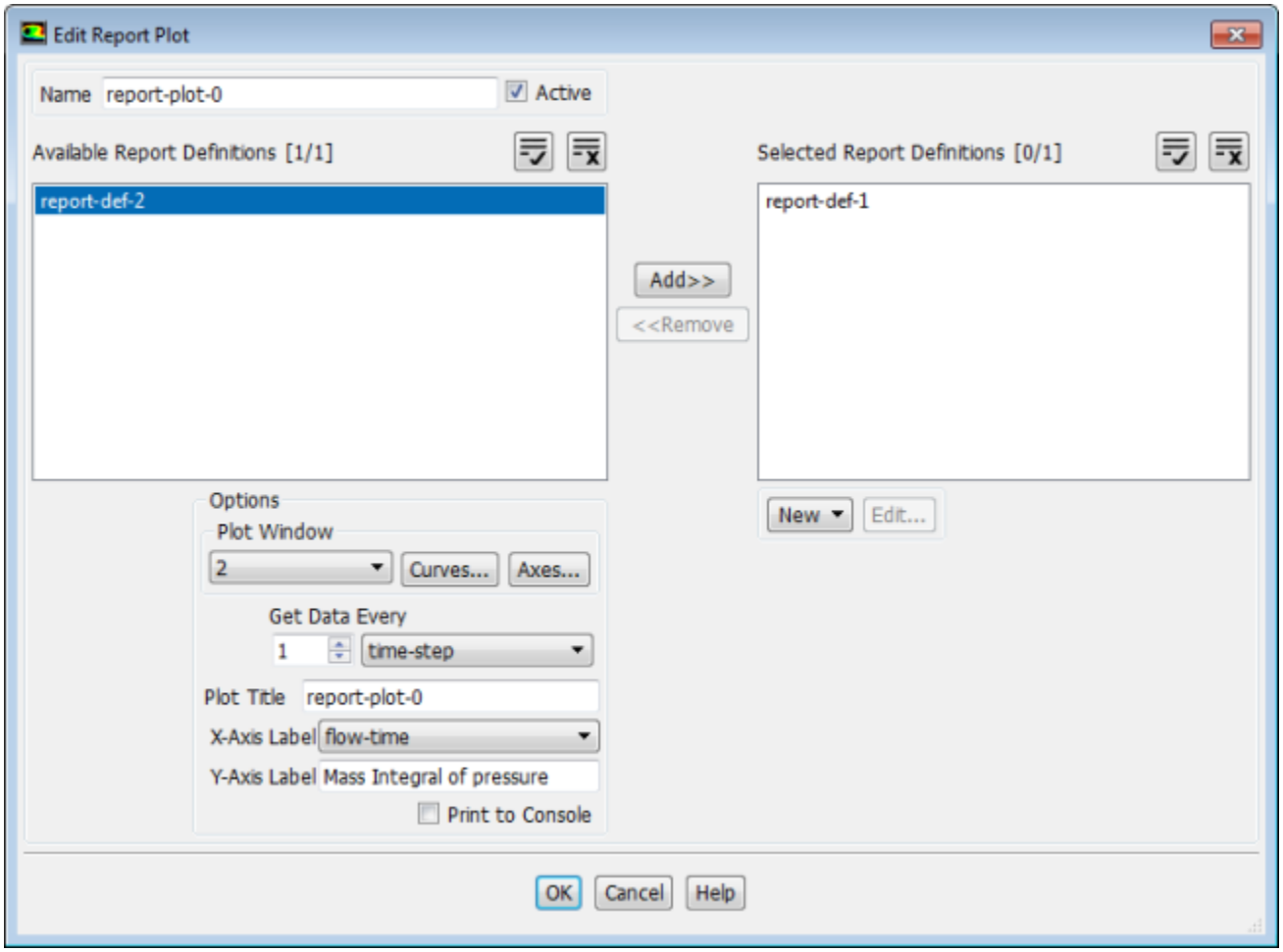
Figure 32.16: Report Plot Definitions Dialog Box



Selecting a report plot to modify opens the Figure 32.17: Edit Report Plot Dialog Box (p. 1909).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 32.17: Edit Report Plot Dialog Box



32.3. Creating Output Parameters

You can create output parameters, which allow you to compare reporting values for different cases, or include reporting values in the function minimized by the mesh morpher/optimizer. These are single values generated by a variety of reports:

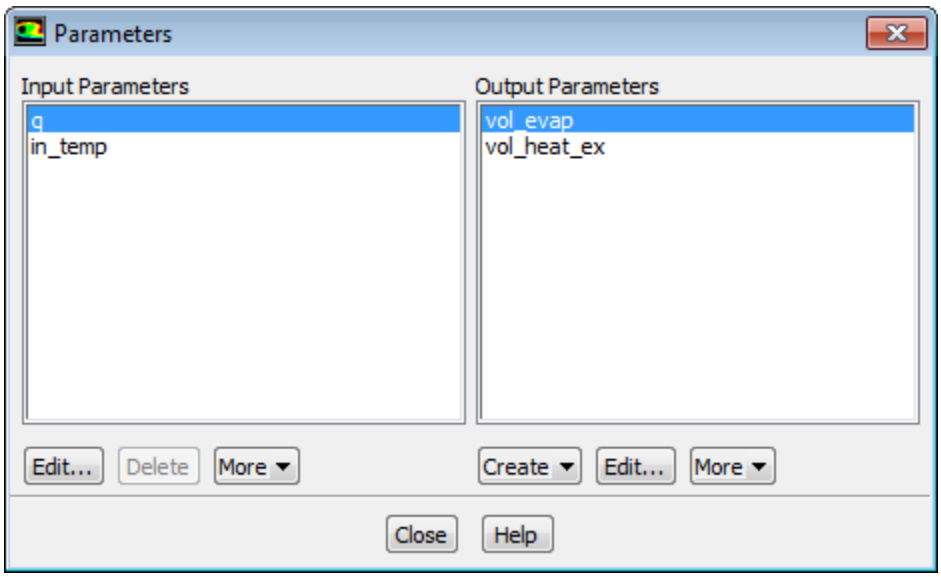
- Fluxes (Fluxes Through Boundaries (p. 1911))
- Forces (Forces on Boundaries (p. 1916))
- Surface integrals (Generating a Surface Integral Report (p. 1920))
- Volume integrals (Generating a Volume Integral Report (p. 1922))
- Surface report definitions (Surface Report definitions (p. 1889))
- Volume report definitions (Volume Report definitions (p. 1891))
- Drag report definitions (Force and Moment Report definitions (p. 1893))
- Lift report definitions(Force and Moment Report definitions (p. 1893))
- Moment report definitions(Force and Moment Report definitions (p. 1893))

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- User Defined report definitions(User Defined Report Definition (p. 1901))
- Expression report definitions (Expression Report Definition (p. 1902))

Click **Parameters...** in the **User Defined** ribbon tab (**Field Functions** group box) to open the Parameters Dialog Box (p. 2627), where a list of any previously created input parameters is available.

User Defined → Field Functions → Parameters...



The list of **Input Parameters** is populated after performing the steps outlined in Defining and Viewing Parameters (p. 243). The output parameters that you create will be listed under **Output Parameters**.

You can define the output parameters using the reporting dialog boxes, as described in Creating Report Definitions (p. 1886). The reporting dialog boxes are accessible **Definitions** drop-down list in the **Solving** ribbon tab (**Reports** group box). Clicking the **Create** drop-down list under **Output Parameters** in the **Parameters** dialog box allows you to select **From Report Definitions...**, which opens the Figure 32.1: Report Definitions Dialog Box (p. 1889). From there you can create any type of report definition.

Once you have saved your output parameters, you can modify their definitions by selecting the parameter in the **Output Parameters** list and clicking **Edit...**. This will open the report dialog box where you can make your changes.

In addition, you can select any of the following commands under the **More** drop-down list:

Delete
removes the selected output parameter from the list of **Output Parameters**.

Print to Console
reports values to the console window. If you select multiple output parameters, then the output includes values from multiple output parameters.

Print All to Console
outputs the values from all output parameters to the console window.

Write...
allows you to store the output to a file. A dialog box is displayed allowing you to provide a file name.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Write All...
prompts you for a file name and then writes the values for all of the output parameters to a file.

32.4. Fluxes Through Boundaries

This section contains information about generating a flux report. For more background information, see Fluxes Through Boundaries in the Theory Guide .

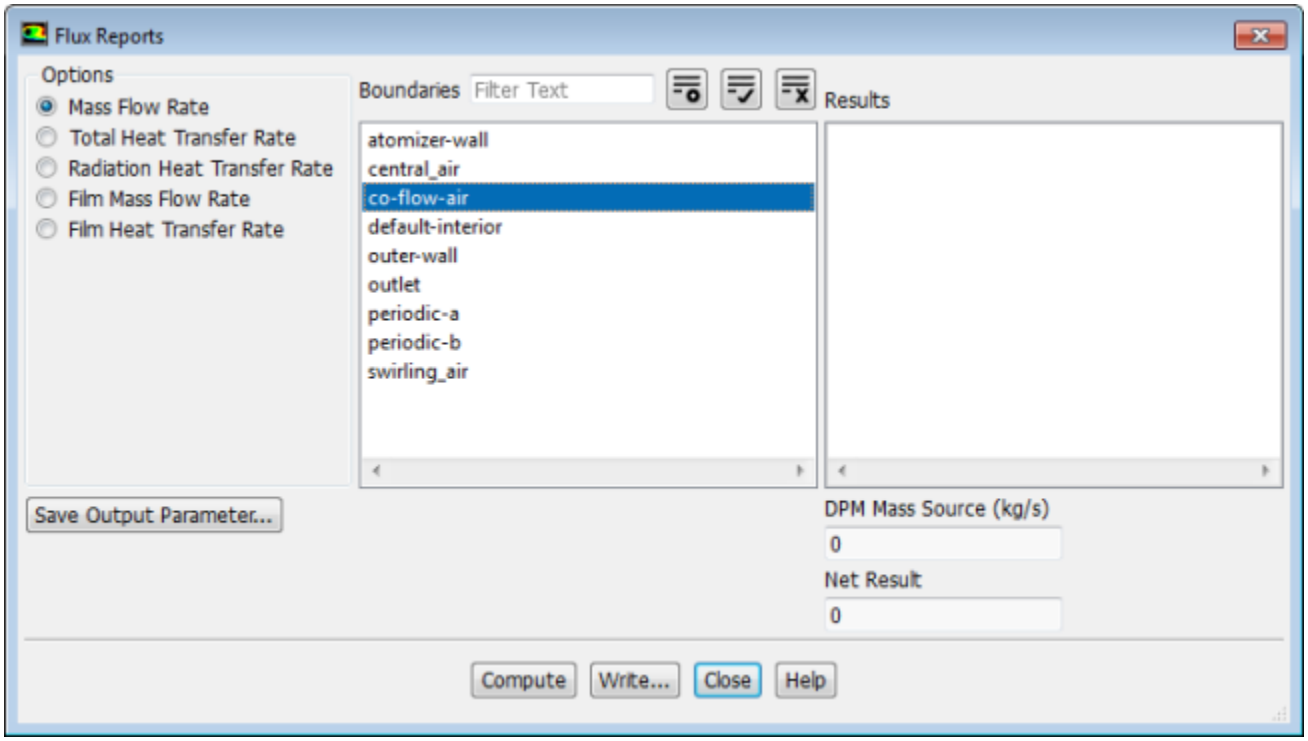
For additional information, see the following sections:
32.4.1. Generating a Flux Report
32.4.2. Flux Reporting for Reacting Flows

32.4.1. Generating a Flux Report

To obtain a report of mass flow rate, total heat transfer rate, total sensible heat transfer rate, or radiation heat transfer rate on selected boundary zones, use the Flux Reports Dialog Box (p. 2613) (Figure 32.18: The Flux Reports Dialog Box (p. 1911)).

Results → Reports → Fluxes Edit...

Figure 32.18: The Flux Reports Dialog Box



The steps for generating the report are as follows:

1. Specify which flux computation you are interested in by selecting one of the following under **Options**:

   - **Mass Flow Rate**
   - **Total Heat Transfer Rate**
   - **Total Sensible Heat Transfer Rate**

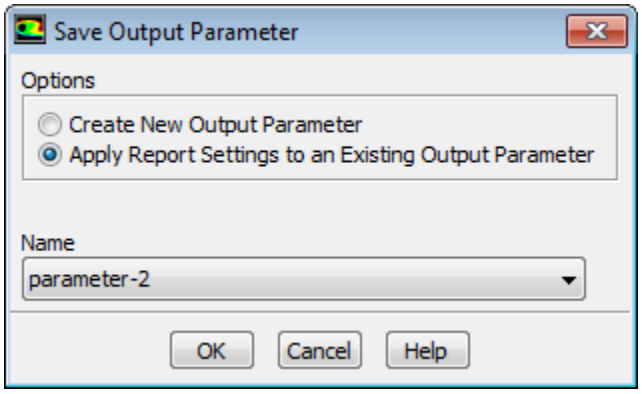
- **Radiation Heat Transfer Rate**
- **Film Mass Flow Rate** (available as an option only when the Eulerian Wall Film model is enabled—see Modeling Eulerian Wall Films (p. 1535))
- **Film Heat Transfer Rate** (available as an option only when the Eulerian Wall Film model is enabled—see Modeling Eulerian Wall Films (p. 1535))

2. In the **Boundaries** list, choose the boundary zone(s) on which you want to report fluxes.

   You can use **Filter Text** entry box to filter the **Boundaries** list to show only the boundary zones that match the pattern you enter. For additional information on using the **Filter Text** entry box, see Filter Text Entry Boxes (p. 15).

3. To create an output parameter for the reported value, click **Save Output Parameter...**. The Save Output Parameter Dialog Box (p. 2631) (Figure 32.19: The Save Output Parameter Dialog Box (p. 1912)) will open where you will specify the name of the newly created output parameter, or overwrite an existing output parameter of the same type. ANSYS Fluent automatically creates generic default names for new input and output parameters (for example, parameter-1, parameter-2, and so on).

   Figure 32.19: The Save Output Parameter Dialog Box



   After the output parameter is created, it is listed in the Parameters Dialog Box (p. 2627), accessed via the **Parameters...** button in the Reports Task Page (p. 2611). You can create any number of output parameters of this report type.

   Note

   If you create an output parameter from a flux report and save the case and data (write), an equivalent report definition will be created when you read in the case file.

4. Click the **Compute** button to display the results of the selected flux computation for each selected boundary zone. The **Net Results** field will show the summation of the individual zone flux results.

Important

Additional steps must be taken prior to generating a flux report for an interior boundary zone that has the same fluid defined on either side. In such a case, the area vectors of the cell faces associated with the zone may have been automatically defined in an inconsistent manner when the mesh file was read into the solver. Since the flux for each individual cell

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

face is calculated with respect to its area vector, such an inconsistency leads to inaccurate results when the face fluxes are summed to calculate the total flux of the boundary zone.

To ensure accurate flux results for such an interior zone, you must orient the area vectors by changing the definition of the zone **Type** to **wall**. You should then change the **Type** back to **interior** and proceed to generate the flux report.

Note that the fluxes are reported exactly as computed by the solver. Therefore, they are inherently more accurate than those computed with the **Flow Rate** option in the Surface Integrals Dialog Box (p. 2616) (described in Surface Integration (p. 1920)).

32.4.2. Flux Reporting for Reacting Flows

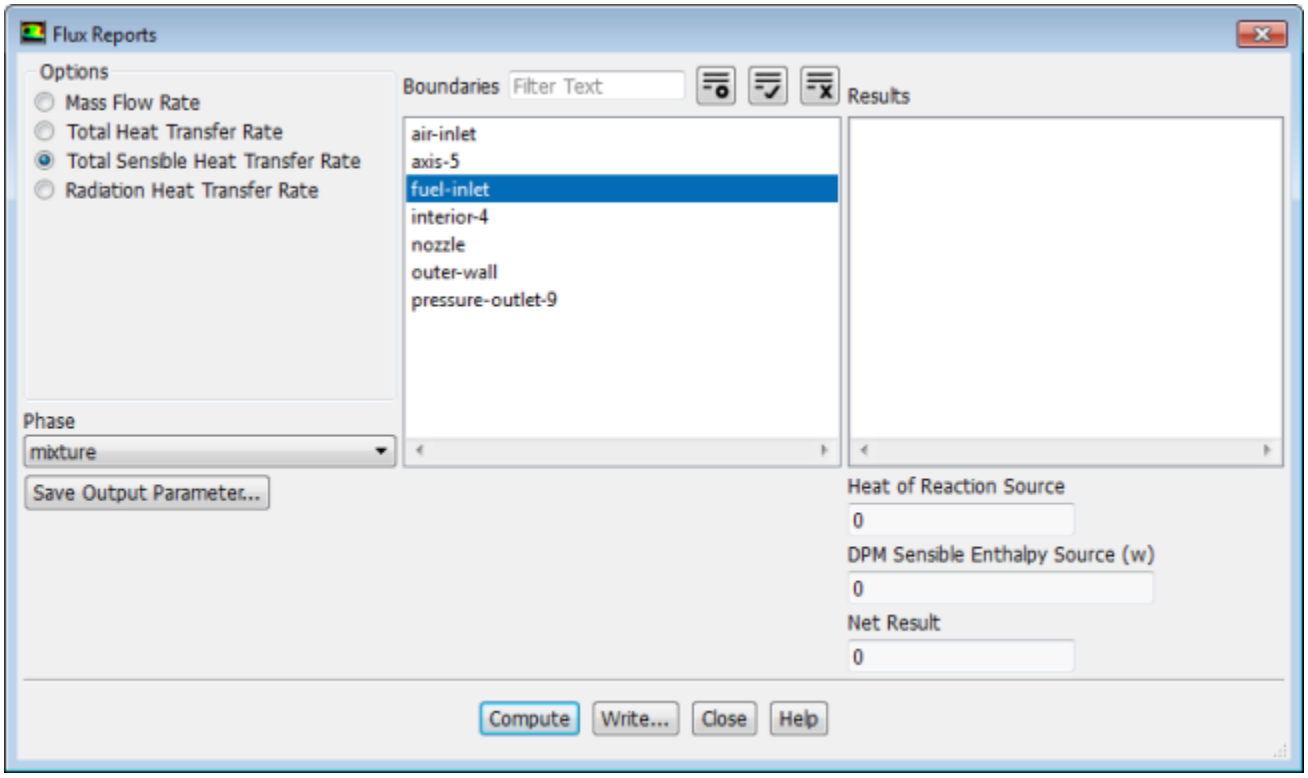
To report heat transfer for reacting flows, one of models in the Species Model Dialog Box (p. 2216) must be enabled for the **Total Sensible Heat Transfer Rate** option to appear in the Flux Reports Dialog Box (p. 2613). For reacting flows, ANSYS Fluent produces two kinds of reports that use a different treatment at the flow boundaries:

- **Total Heat Transfer Rate** reports the total enthalpy flux, which consists of the thermal enthalpy, plus the species formation enthalpy when **Volumetric Reactions** are enabled. The heat rate based on this definition is a conserved quantity in reacting flows. See Heat Transfer Theory in the Theory Guide for details.

- **Total Sensible Heat Transfer Rate** reports the total energy flux as defined in Equation 5.2 in the Theory Guide . Note that in reacting flows, this is not a conserved quantity and the addition or removal of heat due to the chemical reactions (Equation 5.10 in the Theory Guide) is reported separately in the **Heat of Reaction Source** field, as shown in Figure 32.20: The Flux Reports Dialog Box (p. 1914). If you have more than one reaction defined in your case, the **Heat of Reaction Source** reported is the sum of the heat for all reactions. For exothermic reactions the **Heat of Reaction Source** is reported as a positive quantity, while for endothermic reactions it will be a negative quantity.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 32.20: The Flux Reports Dialog Box



Important

Note that both the **Total Heat Transfer Rate** and **Total Sensible Heat Transfer Rate** options report a **Net Result**, which may be used as an indication of the energy balance for the case. In general, and if heat sources other than the heat of reaction and DPM are not included in your problem, the **Net Result** reported in both the **Total Heat Transfer Rate** and **Total Sensible Heat Transfer Rate** options should be a small number for a converged calculation. However, if a reacting case is not well converged for both energy and species transport equations, the **Net Result** reported in the **Total Heat Transfer Rate** and **Total Sensible Heat Transfer Rate** options may differ. In that case, you may consider iterating further to achieve a fully converged solution. In addition, refer to the sections that follow for special considerations when including particles, multiphase models, or other volumetric energy sources.

Important

Note that for the non-premixed and partially premixed models the **Heat of Reaction Source** is calculated as the difference of the net **Total Heat Transfer Rate** and the net **Total Sensible Heat Transfer Rate**. The **Heat of Reaction** field function is not available for the non-premixed and partially-premixed models.

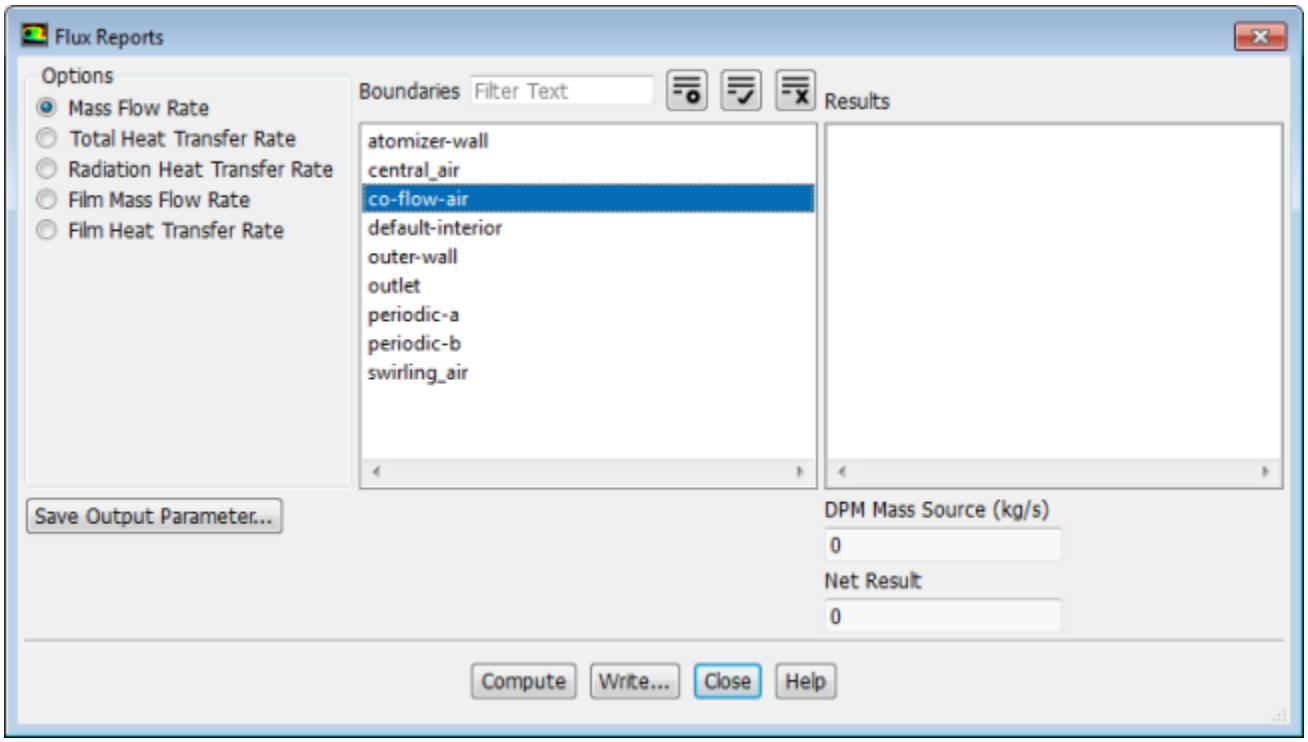
32.4.2.1. Flux Reporting with Particles

If you are using the discrete phase model (DPM), the contributions from the particle injections are reported separately and are included in the net mass and heat balance results. Consequently, the **Mass Flow Rate** report includes the **DPM Mass Source**, the **Total Heat Transfer Rate** report includes the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DPM Enthalpy Source, and the **Total Sensible Heat Transfer Rate** includes the **DPM Sensible Enthalpy Source** (Figure 32.21: The Flux Reports Dialog Box with DPM (p. 1915)).

Figure 32.21: The Flux Reports Dialog Box with DPM



Important

In the case of reacting flows with the DPM model, the **Heat of Reaction Source** entry reports the heat of all homogeneous reactions in the continuous phase, while the heat released or consumed due to particle reactions (e.g, char combustion) is reported in the **DPM Sensible Enthalpy Source** field.

32.4.2.2. Flux Reporting with Multiphase

If you are using any of the multiphase models, the mass or heat rates can be reported separately for each phase and for the mixture phase. Note that if your multiphase model includes mass or heat transfer processes between phases, the mass and heat transferred across the phases will be reported as an imbalance in the report of each phase. In order to check the overall balances for the multiphase cases you should select the mixture phase for your report. In that case, the report will include the sum of the fluxes and sources for all phases included in your model.

Finally, if you are solving a multiphase problem that includes chemical reactions, you should be aware of the following conventions when you are requesting a **Total Sensible Heat Transfer Rate** report:

- If you select one of the phases with gas phase chemical reactions, the **Heat of Reaction Source** will only include contributions from reactions in the particular phase.
- When you report the **Total Sensible Heat Transfer Rate** for the mixture phase, the **Heat of Reaction Source** entry will report the sum of the heat of reaction of all gas phase reactions in all phases plus the heat of any heterogeneous reactions that take place.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.4.2.3. Flux Reporting with Other Volumetric Sources

The reported mass and heat balances address the flow that enters or leaves the domain through boundaries and the contributions from DPM sources; they do not include the contributions from user-defined and other volumetric sources, such as the heat exchanged in the Heat Exchanger Model. For this reason, a mass or heat imbalance may be reported. In that case, and in a converged calculation, the reported imbalance will be equal to the volumetric source.

32.5. Forces on Boundaries

For wall zones that you select, you can compute and report the forces along a specified vector, the moments about a specified center and along a specified axis, and the coordinates of the center of pressure. This feature is useful for reporting, for instance, aerodynamic quantities such as lift, drag, and moment coefficients, as well as the center of pressure for an airfoil.

For additional information about forces, moments, and the center of pressure, see Computing Forces, Moments, and the Center of Pressure in the Theory Guide .

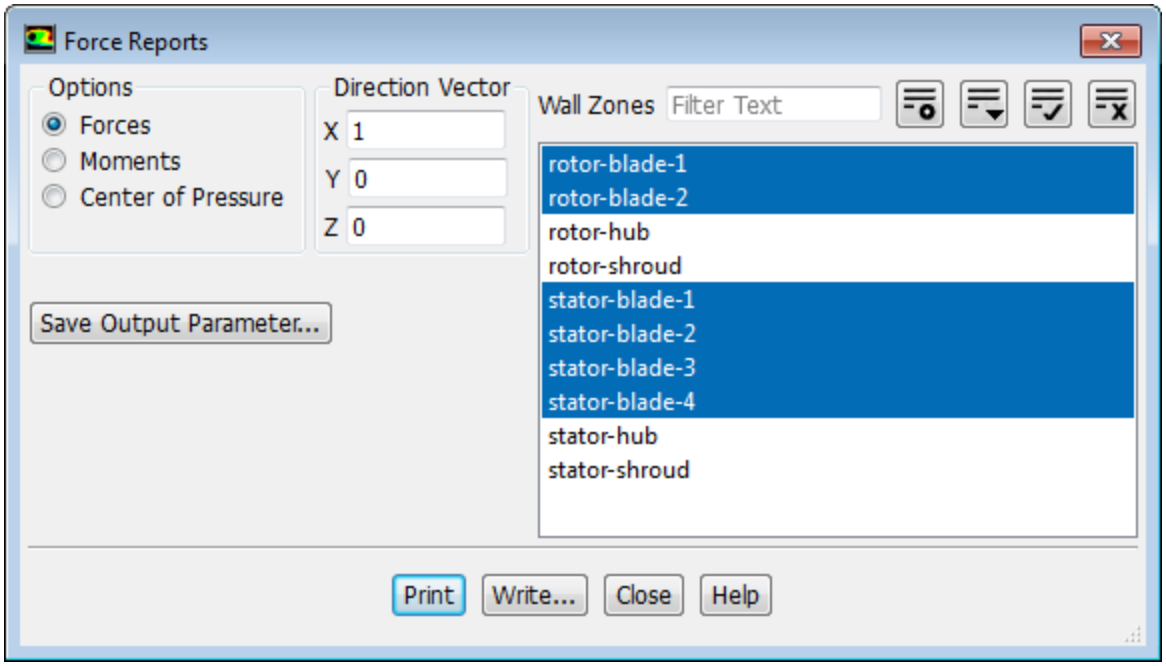
For additional information, see the following section:
32.5.1. Generating a Force, Moment, or Center of Pressure Report

32.5.1. Generating a Force, Moment, or Center of Pressure Report

To obtain a report (for selected wall zones) of forces along a specified vector, moments about a specified center and along a specified axis, or the center of pressure, use the Force Reports Dialog Box (p. 2614) (Figure 32.22: The Force Reports Dialog Box (p. 1916)).

Results → Reports → Forces Edit...

Figure 32.22: The Force Reports Dialog Box



The steps for generating the report are as follows:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Specify the type of report in which you are interested by selecting **Forces**, **Moments**, or **Center of Pressure** from the **Options** list.

2. Define the settings associated with report you are generating:

   a. For a force report, enter the **X**, **Y**, and **Z** components of the **Force Vector** along which the forces will be computed.

   Important

   If F_x (the x-component of the force) is zero, then either the **Y** or **Z** coordinate can be fixed. If F_y is zero, then either the **X** or **Z** coordinate can be fixed. If F_z is zero, then either the **X** or **Y** coordinate can be fixed.

   b. For a moment report, enter the **X**, **Y**, and **Z** coordinates of the **Moment Center** about which the moments will be computed, as well as the **X**, **Y**, and **Z** components of the **Moment Axis** along which the moments will be computed.

   c. For a center of pressure report, define the line (for 2D geometries) or plane (for 3D geometries) on which you want to calculate the center of pressure. The line or plane must have one of its coordinate values fixed (for example, a line defined as $y=10$). Select the axis (**X**, **Y**, or **Z**) in the **Coordinate** group box, and then enter the fixed **Value**. See the example at the end of this section for further details.

3. In the **Wall Zones** list, select the wall zone(s) for which you want a report of the forces, moments, or pressure center.

4. To create an output parameter for the reported value, click **Save Output Parameter...**. The Save Output Parameter Dialog Box (p. 2631) (Figure 32.19: The Save Output Parameter Dialog Box (p. 1912)) will open where you will specify the name of the newly created output parameter, or overwrite an existing output parameter of the same type. ANSYS Fluent automatically creates generic default names for new input and output parameters (for example, parameter-1, parameter-2, and so on).

   After the output parameter is created, it is listed in the Parameters Dialog Box (p. 2627). You can create any number of output parameters of this report type.

   Note

   If you create an output parameter from a force, moment, or center of pressure report and save the case and data (write), an equivalent report definition will be created when you read in the case file.

5. Click the **Print** button if you want the results displayed in the console window, or click **Write...** to save it to a file.

   If you selected **Forces** under **Options**, the pressure force, viscous force (if appropriate), total forces, pressure coefficient, viscous coefficient, and total coefficients for each selected wall zone will be displayed or saved.

   If you selected **Moments**, the pressure moments, viscous moments (if appropriate), total moments, pressure coefficient, viscous coefficient and total coefficients for the wall zones about the specified center will be displayed or saved. Additionally, the moments and coefficients in the direction of the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

specified axis will be displayed or saved. The report will include the values for the individual wall zones, as well as the net values for all of the wall zones combined. See Computing Forces, Moments, and the Center of Pressure in the Theory Guide for details about computing forces and moments.

If you selected **Center of Pressure**, then ANSYS Fluent displays or saves the coordinates about which the moment is zero.

Important

You cannot save your output parameter if **Center of Pressure** is selected; **Center of Pressure** is not available as an output parameter since it is a set of coordinates.

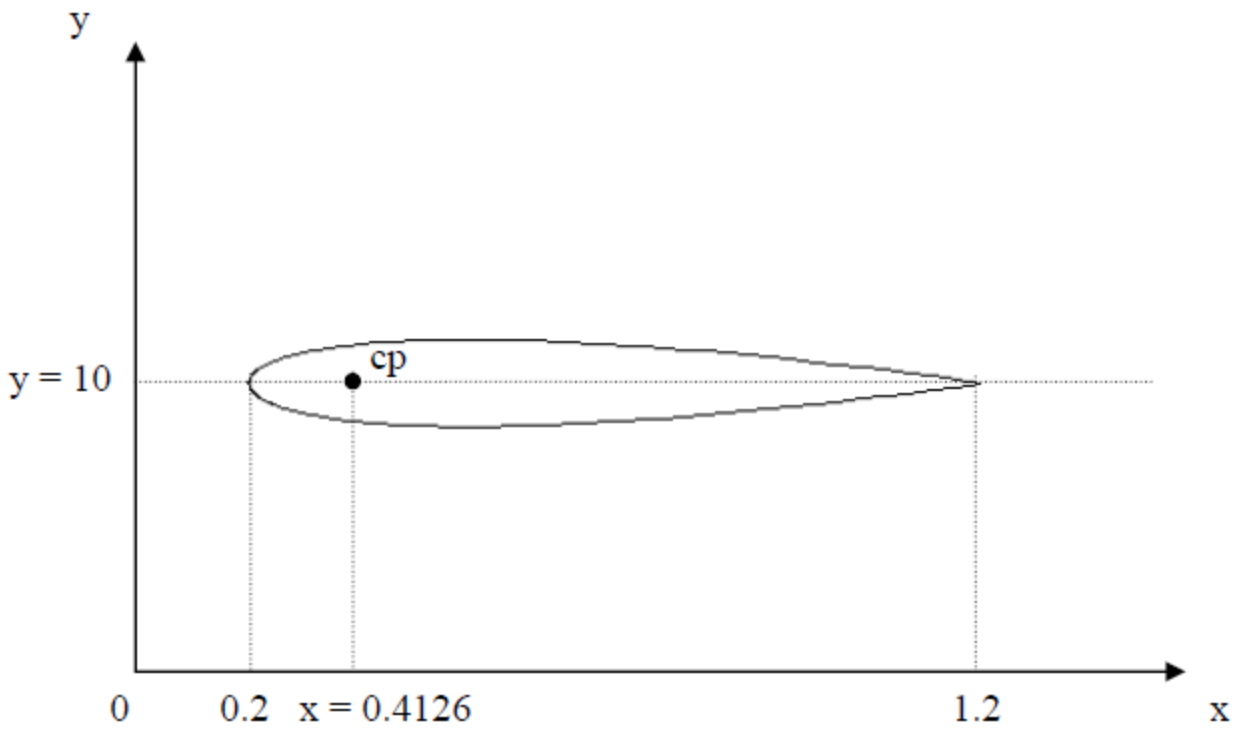
Important

Note that the reported force and moment coefficients are a function of the values entered in the **Reference Values** task page (as described in Computing Forces, Moments, and the Center of Pressure in the Theory Guide). Therefore, appropriate values must be entered in the **Reference Values** task page to get meaningful results.

32.5.1.1. Example

To demonstrate how you would generate and interpret the center of pressure report, consider an airfoil of chord length 1 m (shown in Figure 32.23: An Airfoil with its Computed Center of Pressure (p. 1918)).

Figure 32.23: An Airfoil with its Computed Center of Pressure

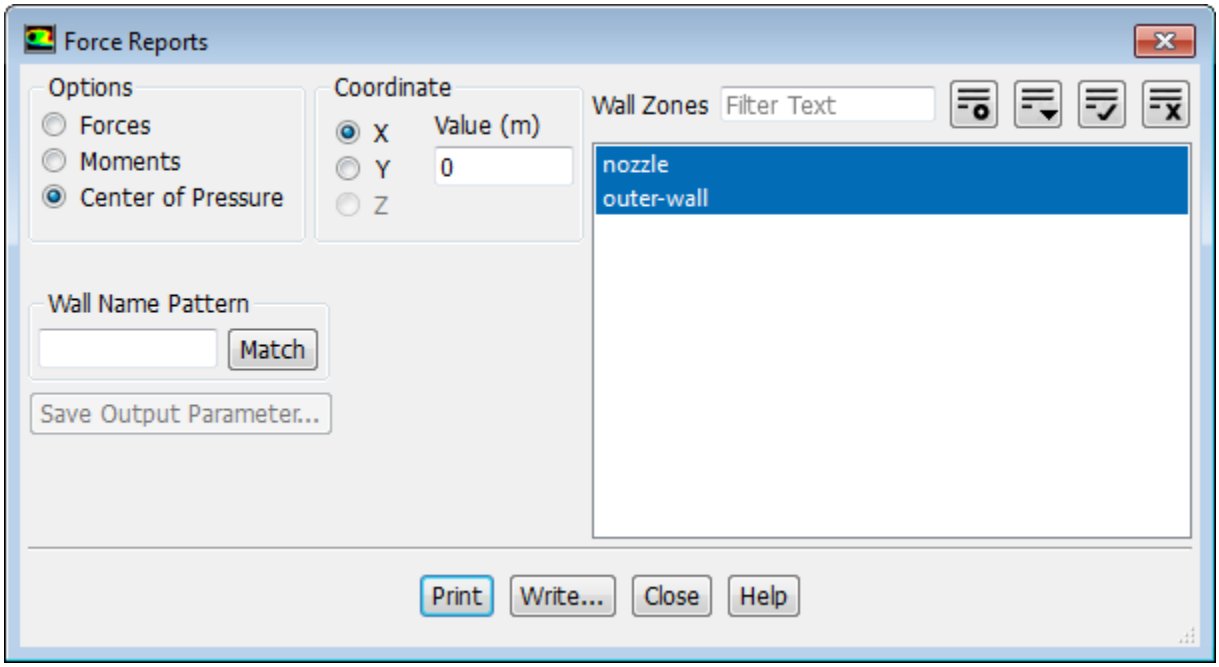


Open the **Force Reports** dialog box and perform the steps that follow.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Results → Reports → Forces Edit...

Figure 32.24: The Force Reports Dialog Box for a Center of Pressure Report



1. Select **Center of Pressure** from the **Options** list.
2. Define the line on which the center of pressure will be calculated. In this case, the **Y** coordinate for the line has a fixed **Value** of `10`.
3. Select the **Wall Zones** that are relevant for the computation.
4. Click **Print** to have the coordinates of the center of pressure displayed in the console window.

The report generated will be in the following form:

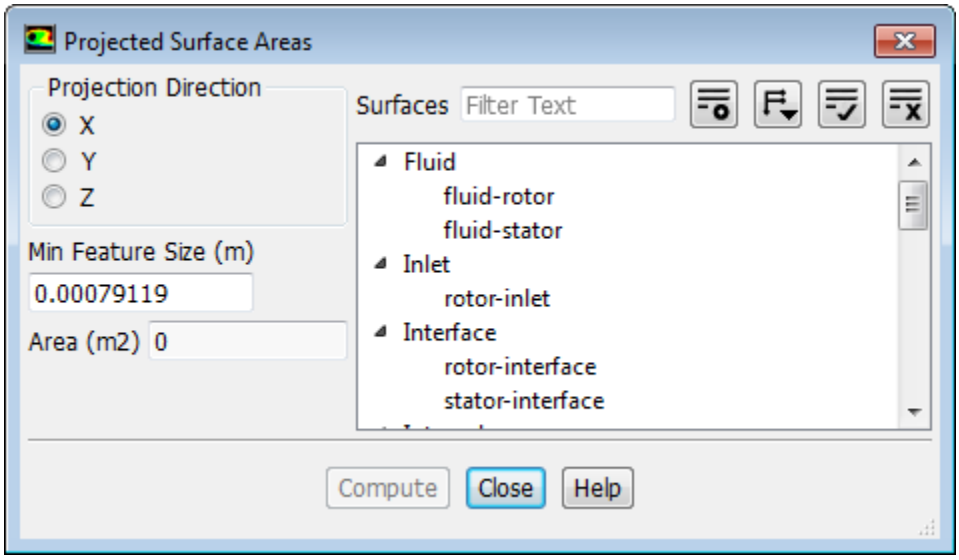
```
Pressure Center Coordinates (in m):
X = 0.41267981
Y = 10
```

32.6. Projected Surface Area Calculations

You can use the Projected Surface Areas Dialog Box (p. 2615) (Figure 32.25: The Projected Surface Areas Dialog Box (p. 1920)) to compute an estimated area of the projection of selected surfaces along the x, y, or z axis (that is, onto the yz, xz, or xy plane).

Results → Reports → Projected Areas Edit...

Figure 32.25: The Projected Surface Areas Dialog Box



The steps for calculating the projected area are as follows:

1. Select the **Projection Direction** (**X**, **Y**, or **Z**).
2. Choose the surface(s) for which the projected area is to be calculated in the **Surfaces** list.
3. Set the **Min Feature Size** to the length of the smallest feature in the geometry that you want to resolve in the area calculation. (You can just use the default value to start with, if you are not sure of the size of the smallest geometrical feature.)
4. Click **Compute**. The area will be displayed in the **Area** box and in the console window.
5. To improve the accuracy of the area calculation, reduce the **Min Feature Size** by half and recompute the area. Repeat this step until the computed **Area** stops changing (or you reach memory capacity).

This feature is available only for 3D domains.

32.7. Surface Integration

This section describes how to compute surface integrals. For mathematical definitions of the various integral types, refer to Computing Surface Integrals in the Theory Guide .

For additional information, see the following section:
32.7.1. Generating a Surface Integral Report

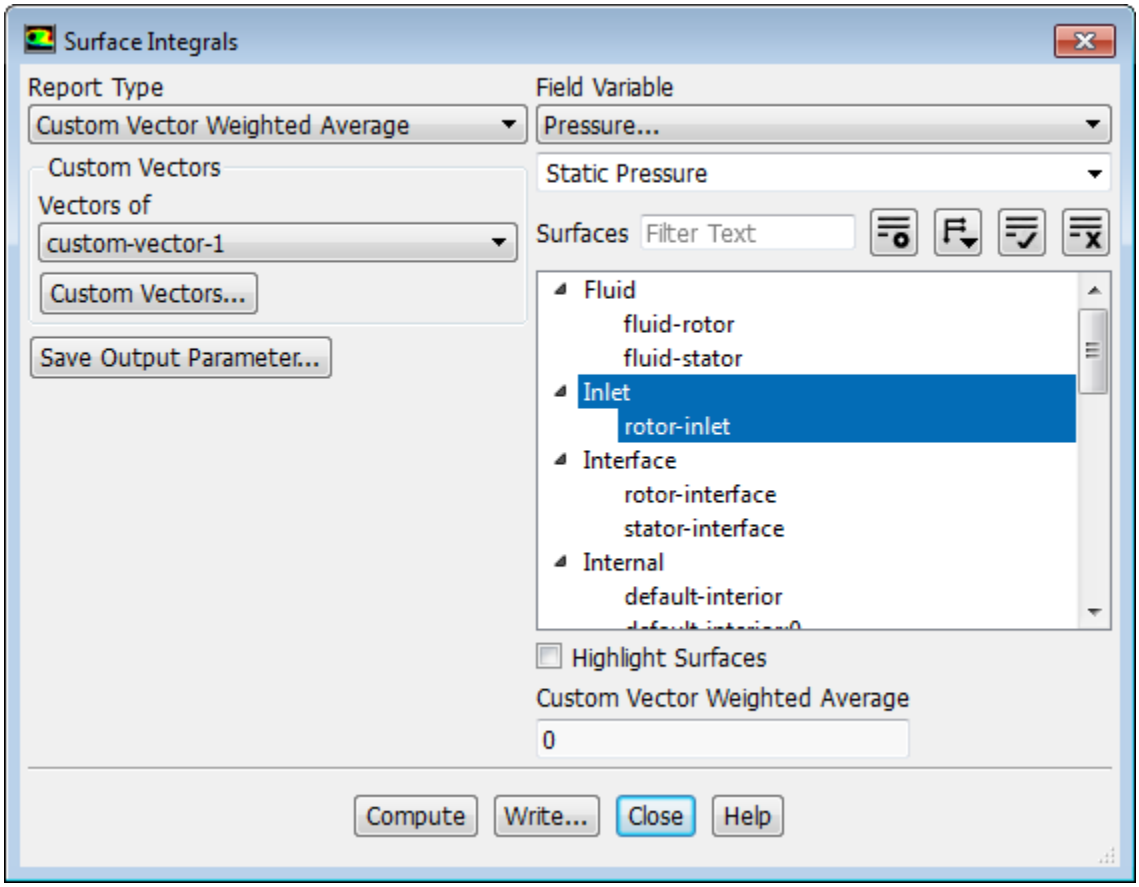
32.7.1. Generating a Surface Integral Report

To obtain a report for selected surfaces of the area, custom vector-based flux, custom vector flux, custom vector-weighted average, mass flow rate, or volume flow rate, or the integral, flow rate, standard deviation, sum, facet maximum, facet minimum, uniformity index (weighted by mass or area), vertex maximum, vertex minimum, or mass-, area-, facet-, or vertex-averaged quantity of a specified field variable, use the Surface Integrals Dialog Box (p. 2616) (Figure 32.26: The Surface Integrals Dialog Box (p. 1921)).

Results → Reports → Surface Integrals Edit...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 32.26: The Surface Integrals Dialog Box



The steps for generating the report are as follows:

1. Specify which type of report you are interested in by selecting **Area**, **Area-Weighted Average**, **Custom Vector Based Flux**, **Custom Vector Flux**, **Custom Vector Weighted Average**, **Facet Average**, **Facet Minimum**, **Facet Maximum**, **Flow Rate**, **Integral**, **Mass Flow Rate**, **Mass-Weighted Average**, **Standard Deviation**, **Sum**, **Uniformity Index - Mass Weighted**, **Uniformity Index - Area Weighted**, **Vertex Average**, **Vertex Minimum**, **Vertex Maximum**, or **Volume Flow Rate** in the **Report Type** drop-down list.

2. If you selected **Custom Vector Based Flux**, **Custom Vector Flux**, or **Custom Vector Weighted Average** from the **Report Type** drop-down list, you must define vectors in the **Custom Vectors** group box.

3. If you are performing a multiphase simulation, you may need to select the phase of interest from the **Phase** drop-down list depending on the selected **Report Type**.

4. If you are generating a report of area, mass flow rate, or volume flow rate, skip to the next step. Otherwise, use the **Field Variable** drop-down lists to select the field variable to be used in the surface integrations. First, select the desired category in the upper drop-down list. You can then select a related quantity from the lower list. (See Field Function Definitions (p. 1931) for an explanation of the variables in the list.)

5. In the **Surfaces** list, choose the surface(s) on which to perform the surface integration.

6. To create an output parameter for the reported value, click **Save Output Parameter...**. The Save Output Parameter Dialog Box (p. 2631) (Figure 32.19: The Save Output Parameter Dialog Box (p. 1912)) will open where you will specify the name of the newly created output parameter, or overwrite an existing output parameter

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of the same type. ANSYS Fluent automatically creates generic default names for new input and output parameters (for example, parameter-1, parameter-2, and so on).

After the output parameter is created, it is listed in the Parameters Dialog Box (p. 2627). You can create any number of output parameters of this report type.

Note

If you create an output parameter from a surface integral report and save the case and data (write), an equivalent report definition will be created when you read in the case file.

---

7. Click the **Compute** button. The computed results are printed in the numeric result field and also in the ANSYS Fluent console.

   The **Net** *value reported in the ANSYS Fluent console corresponds to the specified calculation performed over the list of selected surfaces.*

8. To save the computed results to a file, click the **Write...** button and specify the filename in the resulting **Select File** dialog box.

Note the following:

- Mass averaging "weights" toward regions of higher velocity (that is, regions where more mass crosses the surface).
- Flow rates reported using the Surface Integrals Dialog Box (p. 2616) are not as accurate as those reported with the Flux Reports Dialog Box (p. 2613) (described in Fluxes Through Boundaries (p. 1911)).
- The facet and vertex average options are recommended for zero-area surfaces.
- The uniformity index represents how a specified field variable varies over a surface, where a value of 1 indicates the highest uniformity. The uniformity index can be weighted by area or mass: the area-weighted uniformity index captures the variation of the quantity (for example, the species concentration), whereas the mass-weighted uniformity index captures the variation of the flux (for example, the species flux). See Computing Surface Integrals in the Theory Guide for the equations used to calculate the uniformity index.

32.8. Volume Integration

This section describes how to compute volume integrals. For mathematical definitions of the various integral types, refer to Computing Volume Integrals in the Theory Guide .

For additional information, see the following section:
32.8.1. Generating a Volume Integral Report

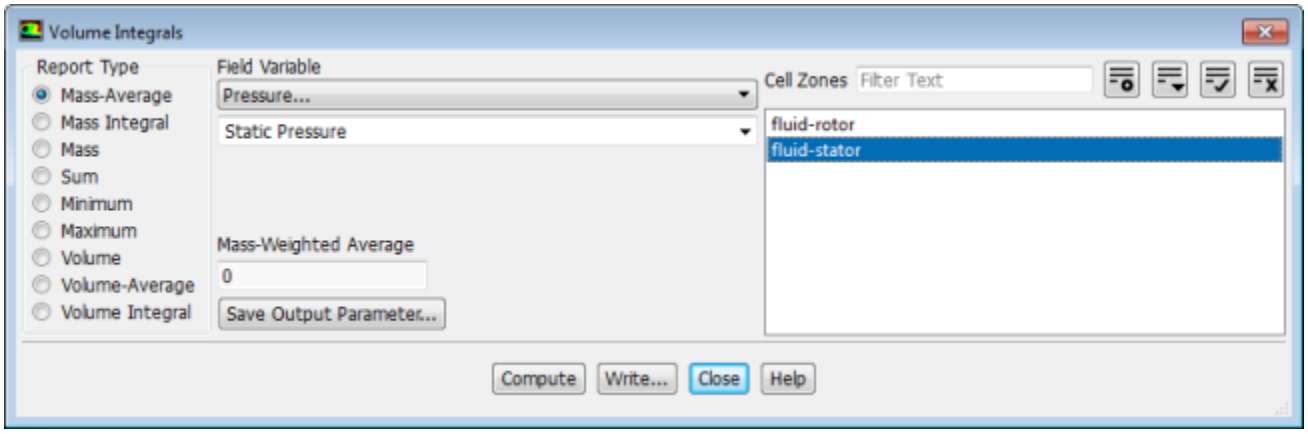
32.8.1. Generating a Volume Integral Report

To obtain a report (of quantities such as the volume, sum, minimum, maximum, volume integral, volume-weighted average, mass-weighted integral, or mass-weighted average) for selected cell zones for a specified field variable, use the Volume Integrals Dialog Box (p. 2620) (Figure 32.27: The Volume Integrals Dialog Box (p. 1923)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Results → Reports → Volume Integrals Edit...

Figure 32.27: The Volume Integrals Dialog Box



The steps for generating the report are as follows:

1. Specify which type of report you are interested in by selecting **Volume**, **Sum**, **Sum*2Pi** (only available for 2D Axisymmetric cases), **Maximum**, **Minimum**, **Volume Integral**, **Volume-Average**, **Mass**, **Mass Integral**, or **Mass-Average** under **Report Type**.

2. If you are generating a report of **Volume** or **Mass**, skip to the next step. Otherwise, use the **Field Variable** drop-down lists to select the field variable to be used in the integral, sum, or averaged volume integrations. First, select the desired category in the upper drop-down list. You can then select a related quantity from the lower list. (See Field Function Definitions (p. 1931) for an explanation of the variables in the list.)

3. If you are performing a multiphase simulation, select the phase of interest (or **mixture**) from the **Phase** drop-down list.

4. In the **Cell Zones** list, choose the zones on which to compute the volume, sum, max, min, volume integral, volume-weighted average, mass integral, or mass-averaged quantity.

5. To create an output parameter for the reported value, click **Save Output Parameter...**. The Save Output Parameter Dialog Box (p. 2631) (Figure 32.19: The Save Output Parameter Dialog Box (p. 1912)) will open where you will specify the name of the newly created output parameter, or overwrite an existing output parameter of the same type. ANSYS Fluent automatically creates generic default names for new input and output parameters (for example, parameter-1, parameter-2, and so on).

   After the output parameter is created, it is listed in the Parameters Dialog Box (p. 2627). You can create any number of output parameters of this report type.

   Note

   If you create an output parameter from a volume integral report and save the case and data (write), an equivalent report definition will be created when you read in the case file.

6. Click the **Compute** button. The computed results are printed in the numeric result field and also in the ANSYS Fluent console.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Net** *value reported in the ANSYS Fluent console corresponds to the specified calculation performed over the list of selected cell zones.*

7. To save the computed results to a file, click the **Write...** button and specify the filename in the resulting **Select File** dialog box.

32.9. Histogram Reports

In ANSYS Fluent, you can print geometric and solution data in the console (text) window in histogram format or plot a histogram in the graphics window. Graphical display of histograms and the procedures for defining a histogram are discussed in Histograms (p. 1851).

The number of cells, the range of the selected variable or function, and the percentage of the total number of cells in the interval will be reported, as in the example below:

```
0 cells below 1.195482 (0 %)
2 cells between 1.195482 and 1.196048 (4.1666667 %)
1 cells between 1.196048 and 1.196614 (2.0833333 %)
0 cells between 1.196614 and 1.19718 (0 %)
0 cells between 1.19718 and 1.197746 (0 %)
2 cells between 1.197746 and 1.198312 (4.1666667 %)
1 cells between 1.198312 and 1.198878 (2.0833333 %)
6 cells between 1.198878 and 1.199444 (12.5 %)
9 cells between 1.199444 and 1.20001 (18.75 %)
25 cells between 1.20001 and 1.200576 (52.083333 %)
2 cells between 1.200576 and 1.201142 (4.1666667
0 cells above 1.201142 (0 %)
```

To generate such a printed histogram, use the Histogram Dialog Box (p. 2598).

Results → Plots → Histogram Edit...

Follow the instructions in Histograms (p. 1851) for generating histogram plots, but click **Print** instead of **Plot** to create the report.

32.10. Discrete Phase

ANSYS Fluent allows you to write particle states (position, velocity, diameter, temperature, and mass flow rate) to files at various boundaries and planes (lines in 2D) using the Sample Trajectories Dialog Box (p. 2622) (Figure 23.47: The Sample Trajectories Dialog Box (p. 1355)). Information about discrete phase reporting is discussed in detail in Sampling of Trajectories (p. 1355), Histogram Reporting of Samples (p. 1356), and Summary Reporting of Current Particles (p. 1358).

32.11. S2S Information

ANSYS Fluent allows you to view the values of the view factor and radiation emitted from one zone to another. You will use the S2S Information Dialog Box (p. 2792) (Figure 12.31: The S2S Information Dialog Box (p. 886)) to generate a report of these values. For details on reporting S2S information, refer to Reporting Radiation in the S2S Model (p. 885).

32.12. Reference Values

You can control the reference values that are used in the computation of derived physical quantities and non-dimensional coefficients. These reference values are used only for postprocessing.

Some examples of the use of reference values include the following:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Force coefficients use the reference area, density, and velocity. In addition, the pressure force calculation uses the reference pressure.
- Moment coefficients use the reference length, area, density and velocity. In addition, the pressure force calculation uses the reference pressure.
- Reynolds number uses the reference length, density, and viscosity.
- Pressure and total pressure coefficients use the reference pressure, density, and velocity.
- Entropy uses the reference density, pressure, and temperature.
- Skin friction coefficient uses the reference density and velocity.
- Heat transfer coefficient uses the reference temperature.
- Turbomachinery efficiency calculations use the ratio of specific heats.

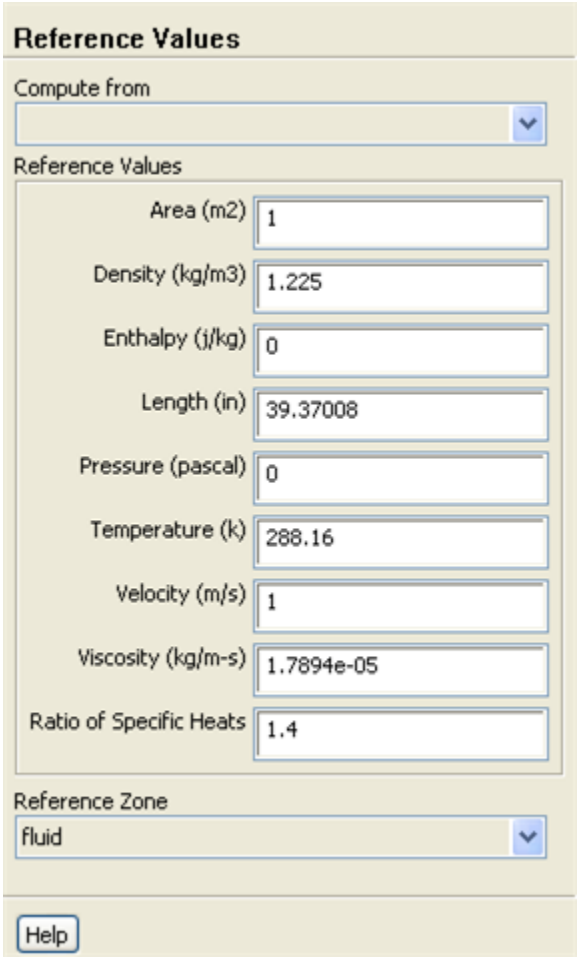
For additional information, see the following sections:
32.12.1. Setting Reference Values
32.12.2. Setting the Reference Zone

32.12.1. Setting Reference Values

To set the reference quantities used for computing normalized flow-field variables, use the Reference Values Task Page (p. 2502) (Figure 32.28: The Reference Values Task Page (p. 1926)).

Setup → Reference Values

Figure 32.28: The Reference Values Task Page



You can specify the reference values manually or compute them based on values of physical quantities at a selected boundary zone. The reference values to be set are **Area**, **Density**, **Enthalpy**, **Length**, **Pressure**, **Temperature**, **Velocity**, dynamic **Viscosity**, and **Ratio Of Specific Heats**.

For 2D problems, an additional quantity, **Depth**, can also be defined. This quantity will be used for reporting fluxes and forces, as well as relevant variables computed using the Surface Integrals Dialog Box (p. 2616) and the Volume Integrals Dialog Box (p. 2620) (for example, **Area**, **Flow Rate**, **Mass Flow Rate**, **Volume**, and so on). You should verify that the value and units of **Depth** corresponds to the depth dimension of your application prior to reporting any of the variables above.

Important

The units for **Depth** are set independently from the units for **Length** in the Set Units Dialog Box (p. 2166).

If you want to compute reference values from the conditions set on a particular boundary zone, select the zone in the **Compute From** drop-down list. Note, however, that depending on the boundary condition used, only some of the reference values may be set. For example, the reference length and area will not be set by computing the reference values from a boundary condition; you will need to set these manually.

To set the values manually, simply enter the value for each under the **Reference Values** heading.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.12.2. Setting the Reference Zone

If you are solving a flow involving multiple reference frames or sliding meshes, you can plot velocities and other related quantities relative to the motion of a specified "reference zone". Choose the desired zone in the **Reference Zone** drop-down list. Changing the reference zone allows you to plot velocities (and total pressure, temperature, and so on) relative to the motion of different zones. See Modeling Flows with Moving Reference Frames (p. 603) for details about postprocessing of relative quantities.

32.13. Summary Reports of Case Settings

You may sometimes find it useful to get a report of the current settings in your case. In ANSYS Fluent, you can list the settings for physical models, boundary conditions, material properties, and solver controls. This report allows you to get an overview of your current problem definition quickly, instead of having to check the settings in each dialog box.

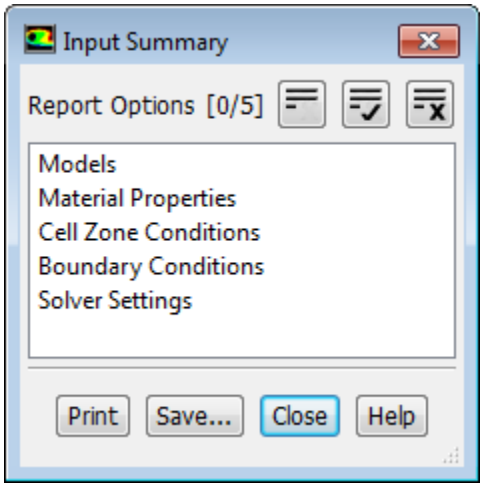
For additional information, see the following section:
32.13.1. Generating a Summary Report

32.13.1. Generating a Summary Report

To generate a summary report you will use the Input Summary Dialog Box (p. 2717) (Figure 32.29: The Input Summary Dialog Box (p. 1927)).

Solving → **Run Calculation** → **Input Summary...**

Figure 32.29: The Input Summary Dialog Box



The steps are as follows:

1. Select the information you would like to see in the report (**Models**, **Boundary Conditions**, **Solver Controls**, and/or **Material Properties**) in the **Report Options** list.

2. To print the information to the ANSYS Fluent console window, click the **Print** button. To save the information to a text file, click the **Save...** button and specify the filename in the resulting **Select File** dialog box.

32.14. System Resource Usage

You can print several reports of memory and processor usage from the **Parallel** ribbon tab.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.14.1. Processor Information
32.14.2. Memory Information
32.14.3. Process and Model Timers

32.14.1. Processor Information

You can print out a table summarizing the processor usage on each machine that has compute processes spawned for the current session by clicking **CPU Info** and selecting **CPU Info** in the **Parallel** ribbon tab (**System** group box). For example, if processes have been spawned on three machines, a table similar to the following is displayed:

```
-------------------------------------------------------------------------------
                   | CPU                                  | System Mem (GB)
Hostname           | Sock x Core  Clock (MHz)  Load       | Total        Available
-------------------------------------------------------------------------------
host23             | 8 x 8, HT    2399.85      0.07       | 129.039      89.565
host24             | 8 x 8, HT    2399.95      1.08       | 129.039      32.516
host25             | 8 x 8, HT    2400.4       3.14       | 129.039      62.531
-------------------------------------------------------------------------------
Total              | 192          -            -          | 387.117      184.612
-------------------------------------------------------------------------------
```

Under CPU:

Sock x Core
displays the number of processor sockets, number of cores per socket, and whether hyper-threading is used.

Clock (MHz)
is the processor speed.

Load
is the work load on the machine.

Under `System Mem (GB)`:

Total
is the total system memory on the machine.

Available
is the available system memory on the machine.

If you are running in parallel, you can print out information about the GPUs available on the machine by clicking **CPU Info** and selecting **GPU Info** in the **Parallel** ribbon tab (**System** group box). If your machine has one or more suitable GPUs present, you can use these to accelerate parallel processing computations as described in Using General Purpose Graphics Processing Units (GPGPUs) With the Algebraic Multigrid (AMG) Solver (p. 2056).

```
CUDA visible GPUs on host01
  CUDA runtime version 5000
  Driver version 6000
  Number of GPUs 1
    0. Quadro K2100M
       3 SMs
       0.6665 GHz
       2.14748 GBytes
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.14.2. Memory Information

You can print out tables summarizing memory usage by node and by host by clicking **CPU Info** and selecting **Memory Usage** in the **Parallel** ribbon tab (**System** group box).

```
-----------------------------------------
       | Virtual Mem Usage (GB)|
ID     | Current      Peak     | Page Faults
-----------------------------------------
host   | 0.0680117    0.0695   | 2.667e+04
n0     | 0.0855195    0.114609 | 7.229e+04
n1     | 0.0869922    0.115367 | 6.988e+04
n2     | 0.0881836    0.117613 | 7.318e+04
n3     | 0.0840781    0.11384  | 6.999e+04
-----------------------------------------
Total  | 0.412785     0.53093  | 3.12e+05
-----------------------------------------

-----------------------------------------------------------------
                   | Virtual Mem Usage (GB)    | System Mem (GB)
Hostname           | Current      Peak         |
-----------------------------------------------------------------
host01             | 0.412785     0.53093      | 32.673
-----------------------------------------------------------------
Total              | 0.412785     0.53093      |
-----------------------------------------------------------------
```

Under `Virtual Mem Usage (GB)`:

Current
is the virtual memory usage at the time the report is generated.

Peak
is the peak virtual memory usage.

(Linux only) Under `Resident Mem Usage (GB)`:

Current
is the resident memory usage at the time the report is generated.

Peak
is the peak resident memory usage.

Page Faults
is the number of page faults that have occurred.

You can use these commands to plan ANSYS Fluent jobs and machines accordingly. They may also be useful to diagnose performance problems.

32.14.3. Process and Model Timers

You can print out detailed information about the CPU timings for the current session as well as solver timings by clicking **CPU Info** and selecting **Time Usage** in the **Parallel** ribbon tab (**System** group box). The solver timings presented are described in Checking Parallel Performance (p. 2060).

```
-----------------------------------------
       | CPU Time Usage (Seconds)
ID     | User         Kernel   Elapsed
-----------------------------------------
host   | 1.34161      0.452403 2328.63
n0     | 3.47882      0.702005 2325.88
n1     | 19.9525      1.15441  2325.87
```

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
n2     | 19.8433      0.873606 2325.87
n3     | 19.8589      0.764405 2325.86
-------------------------------------------
Total  | 64.4752      3.94683  -
-------------------------------------------

Model Timers (Host)
  Other Models Time:                                    2.858 sec
  Total Time:                                           2.858 sec

Model Timers
  Other Models Time:                                    2.871 sec
  Total Time:                                           2.871 sec

Performance Timer for 36 iterations on 4 compute nodes
  Average wall-clock time per iteration:              0.080 sec
  Global reductions per iteration:                       63 ops
  Global reductions time per iteration:               0.000 sec (0.0%)
  Message count per iteration:                         1955 messages
  Data transfer per iteration:                        2.239 MB
  LE solves per iteration:                               4 solves
  LE wall-clock time per iteration:                   0.045 sec (56.2%)
  LE global solves per iteration:                        4 solves
  LE global wall-clock time per iteration:            0.000 sec (0.4%)
  LE global matrix maximum size:                        20
  AMG cycles per iteration:                           8.833 cycles
  Relaxation sweeps per iteration:                      378 sweeps
  Relaxation exchanges per iteration:                   383 exchanges

  Total wall-clock time:                              2.894 sec
  Total CPU time:                                    11.185 sec
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 33: Field Function Definitions

You must select flow variables for a number of tasks in ANSYS Fluent. The values are computed and placed in temporary memory that is allocated for storing the results for each cell. For example, the **Compute** command associated with a dialog box that contains the field variable drop-down list calculates the adjvalues of the selected function and places them into temporary storage.

Node, Cell, and Facet Values (p. 1931) and Velocity Reporting Options (p. 1933) provide some general information related to the field variables. In Field Variables Listed by Category (p. 1934), the variables are listed by category in Table 33.1: **Pressure** and **Density** Categories (p. 1937) – Table 33.17: **Acoustics** Category (p. 1956). These tables will also indicate when each variable will be available. Alphabetical Listing of Field Variables and Their Definitions (p. 1957) contains an alphabetical listing of the variables along with their definitions. All variables appear as they would in the variable selection drop-down lists that are contained in many of the ANSYS Fluent dialog boxes. Custom Field Functions (p. 2003) explains how you can calculate your own field function.

33.1. Node, Cell, and Facet Values
33.2. Velocity Reporting Options
33.3. Field Variables Listed by Category
33.4. Alphabetical Listing of Field Variables and Their Definitions
33.5. Custom Field Functions

33.1. Node, Cell, and Facet Values

For the following discussion, "surface" refers to a collection of facets, lines or points that are created and manipulated in the **Setting Up Domain** ribbon tab. In most cases, these surfaces are created by computing intersections of constant isovalues with the domain cells or with existing surfaces.

For additional information, see the following sections:

33.1.1. Cell Values
33.1.2. Node Values
33.1.3. Facet Values

33.1.1. Cell Values

ANSYS Fluent stores most variables in cells. For postprocessing, the entire region contained within the cell has this value. A surface cell value is the value of the cell that has been intersected by a surface facet or line, or that contains a surface point. Since surface facets and lines are created from the intersection of isovalues and the existing mesh cells, this is a unique definition. Typically, the cell value on a boundary is the value in the cell adjacent to the boundary. For face-only functions like **Wall Shear Stress**, the cell value is the area-weighted average from the face values that define that cell as `c0`. This value is used for the cell values of postprocessing surfaces. But for boundary faces, the cell value actually displays/uses the exact face value.

33.1.2. Node Values

Node values are explicitly defined or obtained by weighted averaging of the cell data. Various boundary conditions impose values of field variables at the domain boundaries, so mesh node values on these

boundary zones are obtained by simple averaging of the adjacent boundary face data. In addition, for several variables (for example, node coordinates) explicit node values are available at all nodes.

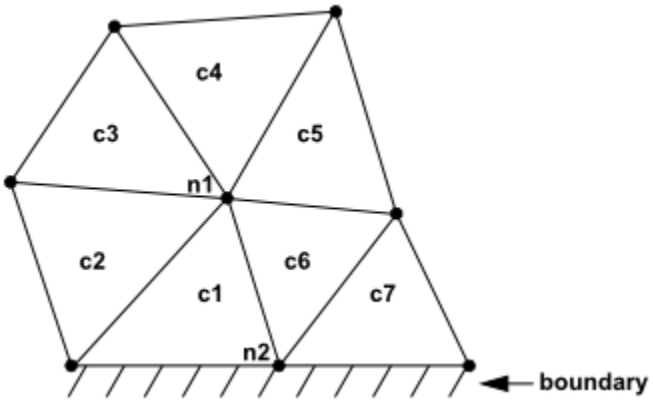
Computation of node values is performed in two steps:

1. Values at all nodes are initialized to the weighted average of the surrounding cell values. The weights are the inverses of the cell volumes that neighbor the nodes.

2. At boundaries, these node values are overwritten with the simple average of the boundary face values. Variables for which explicit node values are available at boundaries are indicated by *bnv* in Table 33.1: **Pressure** and **Density** Categories (p. 1937) – Table 33.17: **Acoustics** Category (p. 1956).

For example, in Figure 33.1: Computing Node Values (p. 1932), the value at node $n1$ will be computed from the weighted average of the values in the surrounding cells ($c1$—$c6$). The value at node $n2$ will be computed from the simple average of the boundary faces ($bf1$ and $bf2$) if there are explicit boundary values available for the variable in question.

Figure 33.1: Computing Node Values



Important

Note that explicit boundary node values are not available for custom field functions.

33.1.2.1. Vertex Values for Points That are Not Mesh Nodes

The values of the nodes on surfaces are linearly interpolated from the mesh node data. For zone surfaces, the nodes on the surface and the zone correspond; therefore, the values are identical. For surfaces that are not zone surfaces (for example, isosurfaces, plane surfaces, and so on), the node values are interpolated from mesh nodes on the cell faces intersected by the postprocessing surface. For point surfaces, the value is interpolated from all the mesh nodes of the cell containing the point.

33.1.3. Facet Values

Facets can be created on preprocessing surfaces and postprocessing surfaces.

33.1.3.1. Facet Values on Zone Surfaces

The interior facets on a zone surface are associated with two cells (`c0` and `c1`). The values of a specified variable on such facets are computed as the arithmetic average of the two cell values of the selected variable.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The boundary facet values of a primary field variables on a zone surface are taken from the specified boundary conditions. The boundary values of dependent variables are computed from the adjacent cell values (versus using the value of the primary variable specified at the boundary). For exact values of dependent variables at the boundaries, define a custom field function specifying the boundary conditions as the primary variables. For more information on custom field functions, see Custom Field Functions (p. 2003).

33.1.3.2. Facet Values on Postprocessing Surfaces

Each facet on a postprocessing surface is associated with a cell. The values of a specified variable on facets are the same as the cell values of the selected variable in the associated cells (this includes iso surfaces, planes, lines, points, rakes, quadric, and so on).

33.2. Velocity Reporting Options

The following methods are available for reporting velocities:

- Cartesian velocities:

  These velocities are based on the Cartesian coordinate system used by the geometry. To report Cartesian velocities, select **X Velocity**, **Y Velocity**, or **Z Velocity**. This is the most common type of velocity reported.

- Cylindrical velocities:

  These velocities are the axial, radial, and tangential components based on the following coordinate systems:

  – For axisymmetric problems, in which the rotation axis must be the x axis, the x direction is the axial direction and the y direction is the radial direction. (If you model axisymmetric swirl, the swirl direction is the tangential direction.)

  – For 2D problems involving a single cell zone, the z direction is the axial direction, and its origin is specified in the Fluid Dialog Box (p. 2369).

  – For 3D problems involving a single cell zone, the coordinate system is defined by the rotation axis and origin specified in the Fluid Dialog Box (p. 2369).

  – For problems involving multiple zones (for example, multiple reference frames or sliding meshes), the coordinate system is defined by the rotation axis specified in the Fluid Dialog Box (p. 2369) (or Solid Dialog Box (p. 2379)) for the "reference zone". The reference zone is chosen in the Reference Values Task Page (p. 2502), as described in Reference Values (p. 1924). Recall that for 2D problems, you will specify only the axis origin; the z direction is always the axial direction.

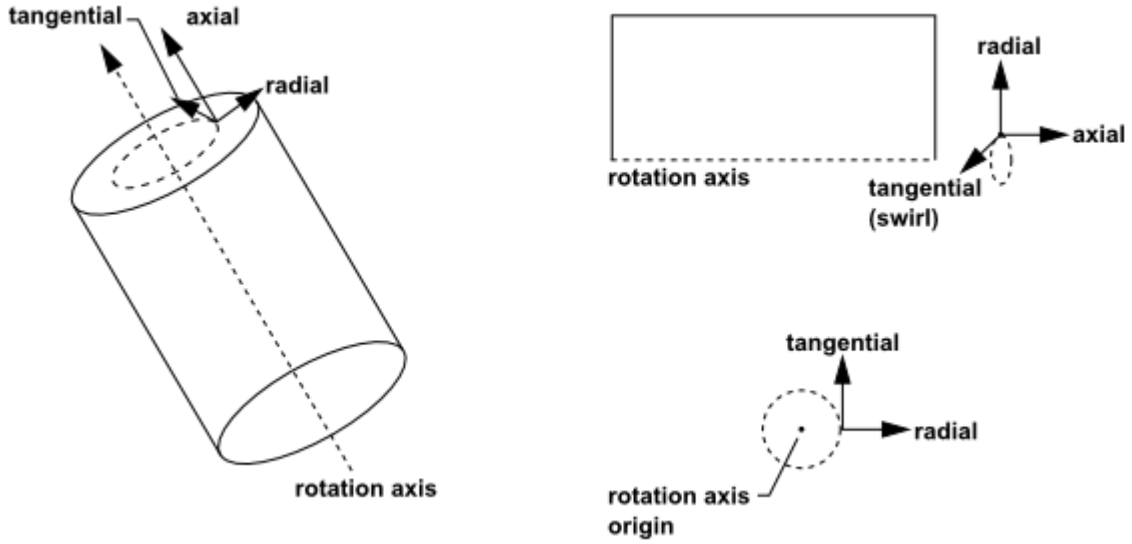
  For all of the above definitions of the cylindrical coordinate system, positive radial velocities point radially out from the rotation axis, positive axial velocities are in the direction of the rotation axis vector, and positive tangential velocities are based on the right-hand rule using the positive rotation axis.

  To report cylindrical velocities, select **Axial Velocity**, **Radial Velocity**, and so on. Figure 33.2: Cylindrical Velocity Components in 3D, 2D, and Axisymmetric Domains (p. 1934) illustrates the cylindrical velocities available for different types of domains. For 3D problems, you can report axial, radial, and tangential velocities. For 2D problems, radial and tangential velocities are available. For axisymmetric problems,

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

you can report axial and radial velocities, and, if you are modeling axisymmetric swirl, you can also report the swirl velocity (which is equivalent to the tangential velocity).

Figure 33.2: Cylindrical Velocity Components in 3D, 2D, and Axisymmetric Domains



- Relative velocities: These velocities are based on the coordinate system and motion of a moving reference frame. They are useful when you are modeling your flow using a moving reference frame, a mixing plane, multiple reference frames, or sliding meshes. (See Modeling Flows with Moving Reference Frames (p. 603) for information about modeling flow in moving zones.) To report relative velocities, select **Relative X Velocity**, **Relative Y Velocity**, **Relative Radial Velocity**, and so on. (Note that you can report relative velocities for both Cartesian and cylindrical components.)

  If you are using a single moving reference frame, the relative velocity values will be reported with respect to the moving frame. If you are using multiple reference frames, mixing planes, or sliding meshes, you will need to specify the frame to which you want the velocities to be relative by choosing the appropriate cell zone as the **Reference Zone** in the Reference Values Task Page (p. 2502) (see Reference Values (p. 1924)). The axis of rotation for each cell zone is defined in the associated Fluid Dialog Box (p. 2369) or Solid Dialog Box (p. 2379). (See Specifying the Rotation Axis (p. 255) or Specifying the Rotation Axis (p. 260) for details.)

  Note that if your problem does not involve any moving zones, relative and absolute velocities will be equivalent.

Note that relative velocities can also be used to compute stagnation quantities (total pressure and total temperature), and that the cylindrical coordinate systems described in the second item above are used for defining the **Axial Coordinate** and **Radial Coordinate** as well.

33.3. Field Variables Listed by Category

In Table 33.1: **Pressure** and **Density** Categories (p. 1937) – Table 33.17: **Acoustics** Category (p. 1956), the following restrictions apply to marked variables:

2d	available only for 2D flows
2da	available only for 2D axisymmetric flows (with or without swirl)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2dasw	available only for 2D axisymmetric swirl flows
3d	available only for 3D flows
adj	available only with data is available from the adjoint solver
ark	available only with the Aungier-Redlich-Kwong real gas model
asfft	available only when created using the Acoustic Sources FFT Dialog Box (p. 2549) or related text commands
bns	available only for broadband noise source models
bnv	node values available at boundaries
cmp	available only for compressible flow
cpl	available only in the density-based solvers
cv	available only for cell values (**Node Values** option turned off)
ddpm	available only when the DDPM model is used
des	available when the turbulence model includes the DES turbulence model
dil	not available with full multicomponent diffusion
do	available only when the discrete ordinates radiation model is used
dpm	available only for coupled discrete phase calculations
dpmean	available only when Mean Values are enabled in the DPM dialog box
dprms	available only when RMS Values are enabled in the DPM dialog box
dtrm	available only when the discrete transfer radiation model is used
fwh	available only with the Ffowcs Williams and Hawkings acoustics model
e	available only for energy calculations
echem	available only with the Electrochemical Reactions model
edc	available only with the EDC model for turbulence-chemistry interaction
emm	available also when the Eulerian multiphase model is used
ewf	available only with the Eulerian Wall Film model
ewt	available only with the enhanced wall treatment
fv	available for face values

gran	available only if a granular phase is present
h2o	available only when the mixture contains water
id	available only when the ideal gas law is enabled for density
ke	available when one of the k-ε turbulence models is used
kklo	available when the k-kl-ω model is used
kw	available when one of the k-ω turbulence models is used
les	available when the LES turbulence model is used
lwf	available only when the Lagrangian wall film model is used
melt	available only when the melting and solidification model is used
mix	available only when the multiphase mixture model is used
mp	available only for multiphase models
netm	available only for porous cell zones with the non-equilibrium thermal model enabled
nox	available only for NOx calculations
np	not available in parallel solvers
nv	uses explicit node value function
os	available only when overset meshing is used
p	available only in parallel solvers
p1	available only when the P-1 radiation model is used
pdf	available only for non-premixed combustion calculations
pmx	available only for premixed combustion calculations
pot	available only when the potential equation is solved
ppmx	available only for partially premixed combustion calculations
r	available only when the Rosseland radiation model is used
rad	available only for radiation heat transfer calculations
rc	available only for finite-rate reactions
rg	available only for the real gas models
rsm	available when the Reynolds stress turbulence model is used

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

s2s	available only when the surface-to-surface radiation model is used
sa	available when the Spalart-Allmaras turbulence model is used
sas	available when the turbulence model includes Scale-Adaptive Simulation (SAS)
sbes	available when the turbulence model includes the Stress-Blended Eddy Simulation (SBES) turbulence model
sdes	available when the turbulence model includes the Shielded Detached Eddy Simulation (SDES) turbulence model
seg	available only in the pressure-based solver
sp	available only for species calculations
sr	available only for surface reactions
sol	available only when the solar model is used
soot	available only for soot calculations
sst	available when the Transition SST model is used
stat	available only with data sampling for transient statistics
stcm	available only for stiff chemistry calculations
t	available only for turbulent flows
th	available only when thin walls are present
turbo	available only when a turbomachinery topology has been defined
udm	available only when a user-defined memory is used
uds	available only when a user-defined scalar is used
v	available only for viscous flows

Table 33.1: Pressure and Density Categories

Category	Variable
Pressure...	**Static Pressure** (bnv) **Pressure Coefficient** **Dynamic Pressure** **Absolute Pressure** (bnv) **Total Pressure** (bnv) **Relative Total Pressure**
Density...	**Density**

Category	Variable
	Density All

Table 33.2: Velocity Category

Category	Variable
Velocity...	**Velocity Magnitude** (bnv)
	X Velocity (bnv)
	Y Velocity (bnv)
	Z Velocity (3d, bnv)
	Swirl Velocity (2dasw, bnv)
	Axial Velocity (2da or 3d)
	Radial Velocity
	Stream Function (2d)
	Tangential Velocity
	Mach Number (id or rg)
	Relative Velocity Magnitude (bnv)
	Relative X Velocity (bnv)
	Relative Y Velocity (bnv)
	Relative Z Velocity (3d, bnv)
	Relative Axial Velocity (2da)
	Relative Radial Velocity (2da)
	Relative Swirl Velocity (2dasw, bnv)
	Relative Tangential Velocity
	Relative Mach Number (id or rg)
	Mesh X-Velocity (nv)
	Mesh Y-Velocity (nv)
	Mesh Z-Velocity (3d, nv)
	Velocity Angle
	Relative Velocity Angle
	Vorticity Magnitude (v)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Helicity (v, 3d)
	X-Vorticity (v, 3d)
	Y-Vorticity (v, 3d)
	Z-Vorticity (v, 3d)
	Cell Reynolds Number (v)
	Cell Convective Courant Number (seg)
	Cell Acoustic Courant Number (cmp, seg)
	Preconditioning Reference Velocity (cpl)

Table 33.3: Temperature, Radiation, and Solidification/Melting Categories

Category	Variable
Temperature...	**Static Temperature** (e, bnv, nv)
	Total Temperature (e, nv)
	Sensible Enthalpy (e, nv)
	Enthalpy (e, nv)
	Relative Total Temperature (e)
	Rothalpy (e, nv)
	Fine Scale Temperature (edc,e)
	Wall Temperature (fv, e, v)
	Wall Temperature (Thin) (th, fv, e, v)
	Total Enthalpy (e)
	Total Enthalpy Deviation (e)
	Entropy (e)
	Total Energy (e)
	Internal Energy (e)
	Non-Equilibrium Thermal Model Source (e, netm)
Radiation...	**Absorption Coefficient** (r, p1, do, or dtrm)
	Scattering Coefficient (r, p1, or do)
	Refractive Index (do)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Radiation Temperature (p1,do, or mc) **Incident Radiation** (p1, do, or mc) **Incident Radiation (Band n)** (p1 (non-gray), do (non-gray), or mc (non-gray)) **Volumetric Absorbed Radiation** (p1, do, or mc) **Volumetric Emitted Radiation** (p1, do, or mc) **Wall Irradiation Flux.Normalized Std Deviation** (mc) **Radiation Intensity.Normalized Std Deviation** (mc) **Surface Cluster ID** (fv, s2s)
Solidification/ Melting	**Liquid Fraction** (melt) **Contact Resistivity** (fv, melt) **X Pull Velocity** (melt (if calculated)) **Y Pull Velocity** (melt (if calculated)) **Z Pull Velocity** (melt (if calculated), 3d) **Axial Pull Velocity** (melt (if calculated), 2da) **Radial Pull Velocity** (melt (if calculated), 2da) **Swirl Pull Velocity** (melt (if calculated), 2dasw)

Table 33.4: Turbulence Category

Category	Variable
Turbulence...	**Turbulent Kinetic Energy (k)** (ke, kw, kklo, sst, rsm, les; sbes, sdes, bnv, nv, or emm) **Laminar Kinetic Energy** (kklo) **Total Fluctuation Energy** (kklo) **Turbulent Intensity** (ke, kw, kklo, sst, rsm, les, sbes, sdes) **Intermittency** (sst, kw, sas, des, sbes, sdes) **Intermittency Effective** (sst, sbes (with Transition SST only), sdes (with Transition SST only))

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Momentum Thickness Re (sst, sbes (with Transition SST only), sdes (with Transition SST only))
	Geometric Roughness Height (sst, sbes (with Transition SST only), sdes (with Transition SST only))
	UU Reynolds Stress (rsm; emm)
	VV Reynolds Stress (rsm; emm)
	WW Reynolds Stress (rsm; emm)
	UV Reynolds Stress (rsm; emm)
	UW Reynolds Stress (rsm, 3d; emm)
	VW Reynolds Stress (rsm, 3d; emm)
	Turbulent Dissipation Rate (Epsilon) (ke, kw, kklo, sst, sbes, sdes, or rsm; bnv (k-epsilon model only), nv (k-epsilon model only), or emm)
	Specific Dissipation Rate (Omega) (kw, kklo, sst, sbes, sdes, rsm)
	Production of k (ke, kw, kklo, sst, rsm, les; emm)
	Production of laminar k (kklo)
	Modified Turbulent Viscosity (sa, des)
	Turbulent Viscosity (sa, ke, kw, kklo, sst, sbes, sdes, rsm, sas, des, les)
	Turbulent Viscosity (large-scale) (kklo)
	Turbulent Viscosity (small-scale) (kklo)
	Effective Viscosity (sa, ke, kw, kklo, sst, sbes, sdes, rsm, des; emm)
	Turbulent Viscosity Ratio (ke, kw, kklo, sst, sbes, sdes, rsm, sa, des, les; emm)
	LES Subgrid Turbulent Viscosity (les)
	Subgrid Kinetic Energy (les)
	Subgrid Turbulent Viscosity (les)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Subgrid Turbulent Viscosity Ratio (les) **Subgrid Effective Viscosity** (les) **Subgrid Filter Length** (les) **Subgrid Test-Filter Length** (les) **Subgrid Dissipation Rate** (les) **Subgrid Dynamic Viscosity Const** (les) **Subgrid Dynamic Prandtl Number** (les) **Subgrid Dynamic Sc of Species** (les) **Subtest Kinetic Energy** (les) **Effective Thermal Conductivity** (t, e) **Effective Prandtl Number** (t, e) **Wall Ystar** (fv, ke, kw, kklo, sst, sbes, sdes, rsm, les) **Wall Yplus** (fv, t) **Turbulent Reynolds Number (Re_y)** (ke, kw, kklo, sst, sbes, sdes, rsm; ewt) **Relative Length Scale (DES)** (des) **DES TKE Dissipation Multiplier** (des) **Curvature Correction Function fr** (sa, ke, kw, sst, sbes, sdes, sas, des) **Shielding Function for SBES or SDES** (sbes, sdes) **Normalized Q criterion** (des, sas, les, sbes, sdes) **Q criterion** (des, sas, les, sbes, sdes)

Table 33.5: Species, Reactions, Pdf, and Premixed Combustion Categories

Category	Variable
Species...	**Mass fraction of species-n** (sp, pdf, or ppmx; nv) **Mole fraction of species-n** (sp, pdf, or ppmx)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Molar Concentration of species-n (sp, pdf, or ppmx)
	Lam Diff Coef of species-n (sp, dil)
	Eff Diff Coef of species-n (t, sp, dil)
	Thermal Diff Coef of species-n (sp)
	Enthalpy of species-n (sp)
	species-n Source Term (rc, cpl)
	Surface Deposition Rate of species-n (sr)
	Surface Coverage of species-n (sr)
	Porous Deposition Rate of species-n (sr)
	Relative Humidity (sp, pdf, or ppmx; h2o)
	Time Step Scale (sp, stcm)
	Fine Scale Mass fraction of species-n (edc)
	Cell Time Scale (edc)
	EDC Cell Volume Fraction (edc)
	DRG Reduced Number of Species (sp)
	Reactor Net Zone ID (sp)
	Reactor Net Temperature (sp)
	Reactor Net Mass fraction of species-n (sp)
	Surface Corrosion Rate of species-n (echem)
Reactions...	**Rate of Reaction-n** (rc)
	Kinetic Rate of Reaction-n (rc)
	Turbulent Rate of Reaction-n (rc, t)
	Heat of Reaction (e, rc)
	Net Reaction Rate of Species-n (edc, stcm)
	DRG Reduced Number of Reactions (sp)
	Echem Reaction Rate of Reaction-n (echem)
Pdf...	**Mean Mixture Fraction** (pdf or ppmx; nv)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Secondary Mean Mixture Fraction (pdf or ppmx; nv) **Mixture Fraction Variance** (pdf or ppmx; nv) **Secondary Mixture Fraction Variance** (pdf or ppmx; nv) **Fvar Prod** (pdf or ppmx) **Fvar2 Prod** (pdf or ppmx) **Scalar Dissipation** (pdf or ppmx) **PDF Table Adiabatic Enthalpy** (pdf or ppmx) **PDF Table Heat Loss/Gain** (e, pdf or ppmx)
Premixed Combustion...	**Progress Variable** (pmx or ppmx; nv) **Damkohler Number** (pmx or ppmx) **Stretch Factor** (pmx or ppmx) **Turbulent Flame Speed** (pmx or ppmx) **Static Temperature** (pmx or ppmx) **Product Formation Rate** (pmx or ppmx) **Laminar Flame Speed** (pmx or ppmx) **Critical Strain Rate** (pmx or ppmx) **Adiabatic Flame Temperature** (pmx or ppmx) **Unburnt Fuel Mass Fraction** (pmx or ppmx)

Table 33.6: NOx, Soot, and Unsteady Statistics Categories

Category	Variable
NOx...	**Mass fraction of NO** (nox) **Mass fraction of HCN** (nox) **Mass fraction of NH3** (nox) **Mass fraction of N2O** (nox) **Mole fraction of NO** (nox) **Mole fraction of HCN** (nox) **Mole fraction of NH3** (nox)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Mole fraction of N2O (nox)
	NO Density (nox)
	HCN Density (nox)
	NH3 Density (nox)
	N2O Density (nox)
	Variance of Temperature (nox)
	Variance of Species (nox)
	Variance of Species 1 (nox)
	Variance of Species 2 (nox)
	Rate of NO (nox)
	Rate of Thermal NO (nox)
	Rate of Prompt NO (nox)
	Rate of Fuel NO (nox)
	Rate of N2OPath NO (nox)
	Rate of Reburn NO (nox)
	Rate of SNCR NO (nox)
	Rate of USER NO (nox)
Soot...	**Average Number of Particles in Soot Aggregate** (soot)
	Mass fraction of soot (soot)
	Mass fraction of Nuclei (soot)
	Mole fraction of soot (soot)
	Normalized Concentration of Nuclei (soot)
	Normalized Soot Aggregation Moments (soot)
	Normalized soot moments (soot)
	Primary Particle Diameter (soot)
	Rate of Coagulation (soot)
	Rate of Nuclei (soot)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Rate of Nucleation (soot)
	Rate of Oxidation (soot)
	Rate of Soot (soot)
	Rate of Soot Mass Nucleation (soot)
	Rate of Surface Growth (soot)
	Soot Density (soot)
	Soot Mean Diameter (soot)
	Soot Surface Area (soot)
	Soot Volume fraction (soot)
Unsteady DPM Statistics...	**Accum DPM Parcels in Cell** (dpm, stat)
	Accum DPM Particles in Cell (dpm, stat)
	Mean DPM *n* (dpm, stat)
	RMS DPM *n* (dpm, stat)
Unsteady Statistics...	**Mean** *n* (stat)
	Mean- *cff_n* (stat)
	RMSE *n* (stat)
	RMSE- *cff_n* (stat)

Table 33.7: Phases, Discrete Phase Model, Granular Pressure, Granular Temperature, and Wall Film Categories

Category	Variable
Phase Interaction...	**Heterogeneous Reaction Rate n**
Phases...	**Volume fraction** (mp)
Discrete Phase Variables...	**DPM Erosion Rate (Generic)** (dpm, cv, fv)
	DPM Erosion Rate (Finnie) (dpm, cv, fv)
	DPM Erosion Rate (McLaury) (dpm, cv, fv)
	DPM Erosion Rate (Oka) (dpm, cv, fv)
	DPM Accretion (dpm, cv, fv)
	DPM Volume Fraction (dpm, dpmean)
	DPM Parcels in Cell (dpm, dpmean)
	DPM Particles in Cell (dpm, dpmean)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	DPM Number Density (dpm, dpmean)
	DPM Collision Rate (dpm)
	DPM X Velocity (dpm, dpmean)
	DPM Y Velocity (dpm, dpmean)
	DPM Z Velocity (dpm, dpmean)
	DPM Diameter (dpm, dpmean)
	DPM Density (dpm, dpmean)
	DPM Temperature (dpm, dpmean)
	DPM Specific Heat (dpm, dpmean)
	DPM D20 (dpm, dpmean)
	DPM D30 (dpm, dpmean)
	DPM D32 (dpm, dpmean)
	DPM Granular Temperature (ddpm, gran)
	DPM Conc of <*component*> (dpm, dpmean, sp)
	DPM RMS X Velocity (dpm, dprms)
	DPM RMS Y Velocity (dpm, dprms)
	DPM RMS Z Velocity (dpm, dprms)
	DPM RMS Diameter (dpm, dprms)
	DPM RMS Temperature (dpm, dprms)
	DPM Absorption Coefficient (dpm, rad)
	DPM Emission (dpm, rad)
	DPM Scattering (dpm, rad)
	DPM Concentration (dpm)
Discrete Phase Sources...	**DPM Mass Source** (dpm)
	DPM X Momentum Source (dpm)
	DPM Y Momentum Source (dpm)
	DPM Z Momentum Source (dpm, 3d)
	DPM Swirl Momentum Source (dpm, 2dasw)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	DPM Sensible Enthalpy Source (dpm, e, rc) **DPM Enthalpy Source** (dpm, e) **DPM Burnout** (dpm, sp, e) **DPM Evaporation/Devolatilization** (dpm, sp, e) **DPM species-n Source** (dpm, sp, e)
Granular Pressure...	**Granular Pressure** (emm, gran)
Granular Temperature...	**Granular Temperature** (emm, gran)
Wall Film...	**Height** (dpm, lwf) **Impingement Mass Flux** (dpm, lwf) **Face Pressure** (dpm, lwf) **Mass** (dpm, lwf) **Temperature** (dpm, lwf) **Heat Flux** (dpm, lwf) **X-Velocity** (dpm, lwf) **Y-Velocity** (dpm, lwf) **Z-Velocity** (dpm, lwf, 3d) **Velocity Magnitude** (dpm, lwf)

Table 33.8: Properties Category

Category	Variable
Properties...	**Molecular Viscosity** (v) **Diameter** (mix, emm) **Granular Conductivity** (mix, emm, gran) **Thermal Conductivity** (e, v) **Specific Heat (Cp)** (e) **Specific Heat Ratio (gamma)** (id) **Gas Constant (R)** (id or rg) **Molecular Prandtl Number** (e, v) **Mean Molecular Weight** (seg, pdf)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Sound Speed (id or rg)
	Compressibility Factor (rg)
	Reduced Temperature (ark)
	Reduced Pressure (ark)
	Critical Temperature (ark, spe)
	Critical Pressure (ark, spe)
	Acentric Factor (ark,spe)
	Critical Specific Volume (ark,spe)
	Spinodal Temperature (ark)
	Electric Conductivity (pot)

Table 33.9: Eulerian Wall Film Category

Category	Variable
Eulerian Wall Film...	**Film Thickness** (3d, ewf)
	Film Mass (3d, ewf)
	Film Temperature (3d, ewf)
	Film X-Velocity (3d, ewf)
	Film Y-Velocity (3d, ewf)
	Film Z-Velocity (3d, ewf)
	Film Velocity Magnitude (3d, ewf)
	Film Effective Pressure (3d, ewf)
	Film Surface X-Velocity (3d, ewf)
	Film Surface Y-Velocity (3d, ewf)
	Film Surface Z-Velocity (3d, ewf)
	Film Surface Velocity Magnitude (3d, ewf)
	Film Surface Temperature (3d, ewf)
	Film Passive Scalar (3d, ewf)
	Film Courant Number (3d, ewf)
	Film Weber Number (3d, ewf)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Film Stripped Mass (3d, ewf) **Film Stripped Diam** (3d, ewf) **Film DPM Mass Source** (3d, ewf, dpm) **Film DPM Energy Source** (3d, ewf, dpm) **Film DPM X-Momentum Source** (3d, ewf, dpm) **Film DPM Y-Momentum Source** (3d, ewf, dpm) **Film DPM Z-Momentum Source** (3d, ewf, dpm) **Film X-Momentum Source** (3d, ewf) **Film Y-Momentum Source** (3d, ewf) **Film Separated Mass** (3d, ewf)

Table 33.10: Sensitivities Category

Category	Variable
Sensitivities...	**Magnitude of Sensitivity to Body Forces (Cell Values)** (adj) **Sensitivity to Body Force X-Component (Cell Values)** (adj) **Sensitivity to Body Force Y-Component (Cell Values)** (adj) **Sensitivity to Body Force Z-Component (Cell Values)** (adj, 3d) **Sensitivity to Mass Sources (Cell Values)** (adj) **Sensitivity to Energy Sources (Cell Values)** (adj, e) **Magnitude of Sensitivity to Body Forces** (adj) **Sensitivity to Body Force X-Component** (adj) **Sensitivity to Body Force Y-Component** (adj) **Sensitivity to Body Force Z-Component** (adj, 3d) **Sensitivity to Mass Sources** (adj)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Sensitivity to Energy Sources (adj, e)
	Sensitivity to Viscosity (adj)
	Shape Sensitivity Magnitude (adj)
	Normal Shape Sensitivity (adj)
	Normal Optimal Displacement (adj)
	log10(Shape Sensitivity Magnitude) (adj)
	Shape Sensitivity X-Component (adj)
	Shape Sensitivity Y-Component (adj)
	Shape Sensitivity Z-Component (adj, 3d)
	Sensitivity to Boundary X-Velocity (adj)
	Sensitivity to Boundary Y-Velocity (adj)
	Sensitivity to Boundary Z-Velocity (adj, 3d)
	Sensitivity to Boundary Pressure (adj)
	Sensitivity to Boundary Temperature (adj, e)
	Sensitivity to Boundary Heat Flux (adj, e)
	Sensitivity to Flow Blockage (adj)
	Adjoint Local Solution Marker (adj)

Table 33.11: Wall Fluxes, User Defined Scalars, and User Defined Memory Categories

Category	Variable
Wall Fluxes...	**Wall Shear Stress** (v, emm, fv)
	X-Wall Shear Stress (v, emm, fv)
	Y-Wall Shear Stress (v, emm, fv)
	Z-Wall Shear Stress (v, 3d, emm, fv)
	Axial-Wall Shear Stress (2da, fv)
	Radial-Wall Shear Stress (2da, fv)
	Swirl-Wall Shear Stress (2dasw, fv)
	Skin Friction Coefficient (v, emm, fv)
	Total Surface Heat Flux (e, v, fv)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Radiation Heat Flux (fv, rad) **Solar Heat Flux** (sol, fv) **Absorbed Radiation Flux (Band-n)** (do, fv) **Absorbed Visible Solar Flux** (sol, fv) **Absorbed IR Solar Flux** (sol, fv) **Reflected Radiation Flux (Band-n)** (do, fv) **Reflected Visible Solar Flux** (sol, fv) **Reflected IR Solar Flux** (sol, fv) **Transmitted Radiation Flux (Band-n)** (do, fv) **Transmitted Visible Solar Flux** (sol, fv) **Transmitted IR Solar Flux** (sol, fv) **Beam Irradiation Flux (Band-n)** (do, fv) **Surface Incident Radiation** (do, dtrm, or s2s; fv) **Surface Heat Transfer Coef.** (e, v, fv) **Wall Func. Heat Tran. Coef.** (e, v, fv) **Surface Nusselt Number** (e, v, fv) **Surface Stanton Number** (e, v, fv)
User Defined Scalars...	**Scalar-n** (uds) **Diffusion Coef. of Scalar-n** (uds)
User Defined Memory...	**udm-n** (udm)

Table 33.12: Cell Info and Mesh Categories

Category	Variable
Cell Info...	**Cell Partition** (np) **Active Cell Partition** (p) **Stored Cell Partition** (p) **Cell Id** (p) **Cell Element Type** **Cell Zone Type**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Cell Zone Index
	Overset Cell Type (os)
	Overset Donor Count (os)
	Overset Receptor Count (os)
	Partition Neighbors
Mesh...	**X-Coordinate** (nv)
	Y-Coordinate (nv)
	Z-Coordinate (3d, nv)
	Axial Coordinate (nv)
	Angular Coordinate (3d, nv)
	Abs. Angular Coordinate (3d, nv)
	Radial Coordinate
	Face Area Magnitude
	X Face Area
	Y Face Area
	Z Face Area (3d)
	Orthogonal Quality
	Cell Equiangle Skew
	Cell Equivolume Skew
	Cell Volume
	2D Cell Volume (2da)
	Interface Overlap Fraction
	Cell Wall Distance
	Face Handedness
	Mark Poor Elements
	Cell Volume Derivative
	Cell Volume Error

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Dynamic Cell Volume

Table 33.13: Mesh Category (Turbomachinery-Specific Variables) and Adaption Category

Category	Variable
Mesh...	**Meridional Coordinate** (nv, turbo) **Abs Meridional Coordinate** (nv, turbo) **Spanwise Coordinate** (nv, turbo) **Abs (H-C) Spanwise Coordinate** (nv, turbo) **Abs (C-H) Spanwise Coordinate** (nv, turbo) **Pitchwise Coordinate** (nv, turbo) **Abs Pitchwise Coordinate** (nv, turbo)
Adaption...	**Adaption Function** **Adaption Curvature** **Adaption Space Gradient** **Adaption Iso-Value** **Existing Value** **Boundary Cell Distance** **Boundary Normal Distance** **Boundary Volume Distance** (np) **Cell Volume Change** **Cell Surface Area** **Cell Warpage** **Cell Children** **Cell Refine**

Table 33.14: Residuals Category

Category	Variable
Residuals...	**Mass Imbalance** (seg) **Pressure Residual** (cpl) **X-Velocity Residual** (cpl) **Y-Velocity Residual** (cpl)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Z-Velocity Residual (cpl, 3d)
	Axial-Velocity Residual (cpl, 2da)
	Radial-Velocity Residual (cpl, 2da)
	Swirl-Velocity Residual (cpl, 2dasw)
	Temperature Residual (cpl, e)
	Species-n Residual (cpl, sp)
	Time Step (cpl)
	Pressure Correction (cpl)
	X-Velocity Correction (cpl)
	Y-Velocity Correction (cpl)
	Z-Velocity Correction (cpl, 3d)
	Axial-Velocity Correction (cpl, 2da)
	Radial-Velocity Correction (cpl, 2da)
	Swirl-Velocity Correction (cpl, 2dasw)
	Temperature Correction (cpl, e)
	Species-n Correction (cpl, sp)

Table 33.15: Derivatives Category

Category	Variable
Derivatives...	**Strain Rate** (v)
	dX-Velocity/dx
	dY-Velocity/dx
	dZ-Velocity/dx (3d)
	dAxial-Velocity/dx (2da)
	dRadial-Velocity/dx (2da)
	dSwirl-Velocity/dx (2dasw)
	d species-n/dx (cpl, sp)
	dX-Velocity/dy
	dY-Velocity/dy

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	dZ-Velocity/dy (3d)
	dAxial-Velocity/dy (2da)
	dRadial-Velocity/dy (2da)
	dSwirl-Velocity/dy (2dasw)
	d species-n/dy (cpl, sp)
	dX-Velocity/dz (3d)
	dY-Velocity/dz (3d)
	dZ-Velocity/dz (3d)
	d species-n/dz (cpl, sp, 3d)
	dOmega/dx (2dasw)
	dOmega/dy (2dasw)
	dp-dX (seg)
	dp-dY (seg)
	dp-dZ (seg, 3d)

Table 33.16: Potential Category

Category	Variable
Potential...	**Electric Potential**
	Current Magnitude
	Joule Heat Source
	Faradaic Current Density (echem)
	Total Surface Corrosion Rate (echem)
	Electrode Surface Potential (echem)
	Faradaic Heat Source (echem)
	Total Echem Heat Source (echem)

Table 33.17: Acoustics Category

Category	Variable
Acoustics...	**Surface dpdt RMS** (fv, fwh)
	Acoustic Power Level (dB) (bns)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Acoustic Power (bns)
	Jet Acoustic Power Level (dB) (bns, 2da)
	Jet Acoustic Power (bns, 2da)
	Surface Acoustic Power Level (dB) (bns, fv)
	Surface Acoustic Power (bns, fv)
	Lilley's Self-Noise Source (bns)
	Lilley's Shear-Noise Source (bns)
	Lilley's Total Noise Source (bns)
	LEE Self-Noise X-Source (bns)
	LEE Shear-Noise X-Source (bns)
	LEE Total Noise X-Source (bns)
	LEE Self-Noise Y-Source (bns)
	LEE Shear-Noise Y-Source (bns)
	LEE Total Noise Y-Source (bns)
	LEE Self-Noise Z-Source (bns, 3d)
	LEE Shear-Noise Z-Source (bns, 3d)
	LEE Total Noise Z-Source (bns, 3d)
	Pressure Spectrum Im *n* (asfft, 3d)
	Pressure Spectrum Re *n* (asfft, 3d)
	SPL for Octave Band at *x* **Hz (dB)** (asfft, 3d)
	SPL for 1/3-Octave Band at *x* **Hz (dB)** (asfft, 3d)
	SPL for Const Width Band *n* **(dB)** (asfft, 3d)

33.4. Alphabetical Listing of Field Variables and Their Definitions

Below, the variables listed in Table 33.1: **Pressure** and **Density** Categories (p. 1937) – Table 33.17: **Acoustics** Category (p. 1956) are defined. For some variables (such as residuals) a general definition is given under the category name, and variables in the category are not listed individually. When appropriate, the unit quantity is included, as it appears in the **Quantities** list in the Set Units Dialog Box (p. 2166).

Abs. Angular Coordinate

(in the **Mesh...** category) is the absolute value of the **Angular Coordinate** defined above.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Abs (C-H) Spanwise Coordinate
(in the **Mesh...** category) is the dimensional coordinate in the spanwise direction, from casing to hub. Its unit quantity is **length**.

Abs (H-C) Spanwise Coordinate
(in the **Mesh...** category) is the dimensional coordinate in the spanwise direction, from hub to casing. Its unit quantity is **length**.

Abs Meridional Coordinate
(in the **Mesh...** category) is the dimensional coordinate that follows the flow path from inlet to outlet. Its unit quantity is **length**.

Abs Pitchwise Coordinate
(in the **Mesh...** category) is the dimensional coordinate in the circumferential (pitchwise) direction. Its unit quantity is **angle**.

Absolute Pressure
(in the **Pressure...** category) is equal to the operating pressure plus the gauge pressure. See Operating Pressure (p. 530) for details. Its unit quantity is **pressure**.

Absorbed Radiation Flux (Band-n)
(in the **Wall Fluxes...** category) is the amount of radiative heat flux absorbed by a semi-transparent wall for a particular band of radiation. Its unit quantity is **heat-flux**.

Absorbed Visible Solar Flux, Absorbed IR Solar Flux
(in the **Wall Fluxes...** category) is the amount of solar heat flux absorbed by a semi-transparent wall or porous jump boundary for a visible or infrared (IR) radiation.

Absorption Coefficient
(in the **Radiation...** category) is the property of a medium that describes the amount of absorption of thermal radiation per unit path length within the medium. It can be interpreted as the inverse of the mean free path that a photon will travel before being absorbed (if the absorption coefficient does not vary along the path). The unit quantity for **Absorption Coefficient** is **length-inverse**.

Accum Parcels in Cell
(in the **Unsteady DPM Statistics...** category) is the cumulative count of parcels passing through a cell since the last reset of the unsteady statistics

Accum Particles in Cell
(in the **Unsteady DPM Statistics...** category) is the cumulative count of particles passing through a cell since the last reset of the unsteady statistics

Acentric Factor
(in the **Properties...** category) is the mixture acentric factor. This property is available when a composition dependent option is selected for acentric factor in the cases with Aungier-Redlich-Kwong real gas model and species transport.

Acoustic Power
(in the **Acoustics...** category) is the acoustic power per unit volume generated by isotropic turbulence (see Equation 15.11 in the Theory Guide). It is available only when the **Broadband Noise Sources** acoustics model is being used. Its unit quantity is **power** per **volume**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Acoustic Power Level (dB)

(in the **Acoustics...** category) is the acoustic power per unit volume generated by isotropic turbulence and reported in dB (see Equation 15.14 in the Theory Guide). It is available only when the **Broadband Noise Sources** acoustics model is being used.

Active Cell Partition

(in the **Cell Info...** category) is an integer identifier designating the partition to which a particular cell belongs. In problems in which the mesh is divided into multiple partitions to be solved on multiple processors using the parallel version of ANSYS Fluent, the partition ID can be used to determine the extent of the various groups of cells. The active cell partition is used for the current calculation, while the stored cell partition (the last partition performed) is used when you save a case file. See Partitioning the Mesh Manually and Balancing the Load (p. 2033) for more information.

Adaption...

includes field variables that are commonly used for adapting the mesh. For information about solution adaption, see Adapting the Mesh (p. 1681).

Adaption Function

(in the **Adaption...** category) can be either the **Adaption Space Gradient** or the **Adaption Curvature**, depending on the settings in the Gradient Adaption Dialog Box (p. 2710). For instance, the **Adaption Curvature** is the undivided Laplacian of the values in temporary cell storage. To display contours of the Laplacian of pressure, for example, you first select **Static Pressure**, click the **Compute** (or **Display**) button, select **Adaption Function**, and finally click the **Display** button.

Adaption Iso-Value

(in the **Adaption...** category) is the desired field variable function.

Adaption Space Gradient

(in the **Adaption...** category) is the first derivative of the desired field variable.

$$|e_{i1}| = \left(A_{cell}\right)^{\frac{r}{2}} |\nabla f| \tag{33.1}$$

Depending on the settings in the Gradient Adaption Dialog Box (p. 2710), this equation will either be scaled or normalized. Recommended for problems with shock waves (that is, supersonic, inviscid flows). For more information, see Gradient Adaption Approach in the Theory Guide .

Adaption Curvature

(in the **Adaption...** category) is the second derivative of the desired field variable.

$$|e_{i2}| = \left(A_{cell}\right)^{\frac{r}{2}} |\nabla^2 f| \tag{33.2}$$

Depending on the settings in the Gradient Adaption Dialog Box (p. 2710), this equation will either be scaled or normalized. Recommended for smooth solutions (that is, viscous, incompressible flows). For more information, see Gradient Adaption Approach in the Theory Guide .

Adiabatic Flame Temperature

(in the **Premixed Combustion...** category) is the adiabatic temperature of burnt products in a laminar premixed flame (T_b in Equation 9.68 in the Theory Guide). Its unit quantity is **temperature**.

Adjoint Local Solution Marker

(in the **Sensitivities...** category), intended for expert users, can be plotted to identify those portions of the flow domain where the stabilized adjoint solution advancement scheme is applied. It is preferable to plot this with the **Node Values** de-selected in the **Contours** dialog box. In this case, the **Adjoint Local**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution Marker will take a value between 0 and 1. The **Mode Amplitude Cutoff** defined in the **Stabilized Scheme Settings** dialog box defines the lower bound for cells where the stabilized scheme is applied.

Angular Coordinate

(in the **Mesh...** category) is the angle between the radial vector and the position vector. The radial vector is obtained by transforming the default radial vector (y-axis) by the same rotation that was applied to the default axial vector (z-axis). This assumes that, after the transformation, the default axial vector (z-axis) becomes the reference axis. The angle is positive in the direction of cross-product between reference axis and radial vector.

Average Number of Particles in Soot Aggregate

(In the **Soot…** category) is an average number of primary particles in a soot aggregate (n_p in equation Equation 14.185 in the *Fluent Theory Guide*). This quantity is available only with the Method of Moments model.

Axial Coordinate

(in the **Mesh...** category) is the distance from the origin in the axial direction. The axis origin and (in 3D) direction is defined for each cell zone in the Fluid Dialog Box (p. 2369) or Solid Dialog Box (p. 2379). The axial direction for a 2D model is always the z direction, and the axial direction for a 2D axisymmetric model is always the x direction. The unit quantity for **Axial Coordinate** is **length**.

Axial Pull Velocity

(in the **Solidification/Melting...** category) is the axial-direction component of the pull velocity for the solid material in a continuous casting process. Its unit quantity is **velocity**.

Axial Velocity

(in the **Velocity...** category) is the component of velocity in the axial direction. (See Velocity Reporting Options (p. 1933) for details.) For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list. Its unit quantity is **velocity**.

Axial-Wall Shear Stress

(in the **Wall Fluxes...** category) is the axial component of the force acting tangential to the surface due to friction. Its unit quantity is **pressure**.

Beam Irradiation Flux (Band-b)

(in the **Wall Fluxes...** category) is specified as an incident heat flux (W / m^2) for each wavelength band.

Boundary Cell Distance

(in the **Adaption...** category) is an integer that indicates the approximate number of cells from a boundary zone.

Boundary Normal Distance

(in the **Adaption...** category) is the distance of the cell centroid from the closest boundary zone.

Boundary Volume Distance

(in the **Adaption...** category) is the cell volume distribution based on the **Boundary Volume**, **Growth Factor**, and normal distance from the selected **Boundary Zones** defined in the Boundary Adaption Dialog Box (p. 2673). See Boundary Adaption (p. 1685) for details.

Cell Children

(in the **Adaption...** category) is a binary identifier based on whether a cell is the product of a cell subdivision in the hanging-node adaption process (value = 1) or not (value = 0).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cell Acoustic Courant Number

(in the **Velocity...** category) is a ratio of the time step size to the acoustic wave propagation time based on the cell size.

Cell Convective Courant Number

(in the **Velocity...** category) is a ratio of the time step size to the convective wave propagation time based on the cell size.

Cell Element Type

(in the **Cell Info...** category) is the integer cell element type identification number. Each cell can have one of the following element types:

```
triangle     1
tetrahedron  2
quadrilateral 3
hexahedron   4
pyramid    5
wedge      6
polyhedra  7
```

Cell Equiangle Skew

(in the **Mesh...** category) is a nondimensional parameter calculated using the normalized angle deviation method, and is defined as

$$\max\left[\frac{q_{max}-q_e}{180-q_e}, \frac{q_e-q_{min}}{q_e}\right] \tag{33.3}$$

where

q_{max} = largest angle in the face or cell

q_{min} = smallest angle in the face or cell

q_e = angle for an equiangular face or cell (for example, 60 for a triangle and 90 for a square)

A value of 0 indicates a best case equiangular cell, and a value of 1 indicates a completely degenerate cell. Degenerate cells (slivers) are characterized by nodes that are nearly coplanar (collinear in 2D). **Cell Equiangle Skew** applies to all elements.

Cell Equivolume Skew

(in the **Mesh...** category) is a nondimensional parameter calculated using the volume deviation method, and is defined as

$$\frac{(\text{optimal-cell-size})-(\text{cell-size})}{(\text{optimal-cell-size})} \tag{33.4}$$

where optimal-cell-size is the size of an equilateral cell with the same circumradius. A value of 0 indicates a best case equilateral cell and a value of 1 indicates a completely degenerate cell. Degenerate cells (slivers) are characterized by nodes that are nearly coplanar (collinear in 2D). **Cell Equivolume Skew** applies only to triangular and tetrahedral elements.

Cell Id

(in the **Cell Info...** category) is a unique integer identifier associated with each cell.

Cell Info...

includes quantities that identify the cell and its relationship to other cells.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cell Partition

(in the **Cell Info...** category) is an integer identifier designating the partition to which a particular cell belongs. In problems in which the mesh is divided into multiple partitions to be solved on multiple processors using the parallel version of ANSYS Fluent, the partition ID can be used to determine the extent of the various groups of cells.

Cell Refine Level

(in the **Adaption...** category) is an integer that indicates the number of times a cell has been subdivided in the hanging node adaption process, compared with the original mesh. For example, if one quad cell is split into four quads, the **Cell Refine Level** for each of the four new quads will be 1. If the resulting four quads are split again, the **Cell Refine Level** for each of the resulting 16 quads will be 2.

Cell Reynolds Number

(in the **Velocity...** category) is the value of the Reynolds number in a cell. (Reynolds number is a dimensionless parameter that is the ratio of inertia forces to viscous forces.) **Cell Reynolds Number** is defined as

$$Re \equiv \frac{\rho u d}{\mu} \tag{33.5}$$

where ρ is density, u is velocity magnitude, μ is the effective viscosity (laminar plus turbulent), and d is **Cell Volume** $^{1/2}$ for 2D cases and **Cell Volume** $^{1/3}$ in 3D or axisymmetric cases.

Cell Surface Area

(in the **Adaption...** category) is the total surface area of the cell, and is computed by summing the area of the faces that compose the cell.

Cell Volume

(in the **Mesh...** category) is the volume of a cell. In 2D the volume is the area of the cell multiplied by the unit depth. For axisymmetric cases, the cell volume is calculated using a reference depth of 1 radian. The unit quantity of **Cell Volume** is **volume**.

Cell Volume Derivative

(in the **Mesh...** category) is the change of a cell volume over time.

Cell Volume Error

(in the **Mesh...** category) is the cell volume over the unsteady cell volume.

2D Cell Volume

(in the **Mesh...** category) is the two-dimensional volume of a cell in an axisymmetric computation. For an axisymmetric computation, the 2D cell volume is scaled by the radius. Its unit quantity is **area**.

Cell Volume Change

(in the **Adaption...** category) is the maximum volume ratio of the current cell and its neighbors.

Cell Wall Distance

(in the **Mesh...** category) is the distribution of the normal distance of each cell centroid from the wall boundaries. Its unit quantity is **length**.

Cell Warpage

(in the **Adaption...** category) is the square root of the ratio of the distance between the cell centroid and cell circumcenter and the circumcenter radius:

$$\text{warpage} = \sqrt{\frac{\left|\vec{r}_{centroid} - \vec{r}_{circumcenter}\right|}{R_{circumcenter}}} \tag{33.6}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cell Zone Index

(in the **Cell Info...** category) is the integer cell zone identification number. In problems that have more than one cell zone, the cell zone ID can be used to identify the various groups of cells.

Cell Zone Type

(in the **Cell Info...** category) is the integer cell zone type ID. A fluid cell has a type ID of 1, a solid cell has a type ID of 17, and an exterior cell (parallel solver) has a type ID of 21.

Compressibility Factor

(in the **Properties...** category) is the ratio of the ideal gas density of the fluid divided by the real gas fluid density in the same flow conditions. Compressibility Factor is defined as

$$Z=\frac{P/RT}{\rho} \tag{33.7}$$

where Z is the compressibility factor, P is the absolute pressure, T is the temperature, and $R=R_u/MW$ (the universal gas constant R_u divided by the molecular weight MW). The compressibility factor is available only with the real gas models.

Contact Resistivity

(in the **Solidification/Melting...** category) is the additional resistance at the wall due to contact resistance. It is equal to $R_c\left(1-\beta\right)/h$, where R_c is the contact resistance, β is the liquid fraction, and h is the cell height of the wall-adjacent cell. The unit quantity for **Contact Resistivity** is **thermal-resistivity**.

Critical Pressure

(in the **Properties...** category) is the mixture critical pressure. This property is available when a composition dependent option is selected for critical pressure in the cases with Aungier-Redlich-Kwong real gas model and species transport.

Critical Specific Volume

(in the **Properties...** category) is the mixture critical specific volume. This property is available when a composition dependent option is selected for critical specific volume in the cases with Aungier-Redlich-Kwong real gas model and species transport.

Critical Strain Rate

(in the **Premixed Combustion...** category) is a parameter that takes into account the stretching and extinction of premixed flames (g_{cr} in Equation 9.18 in the Theory Guide). Its unit quantity is **time-inverse**.

Critical Temperature

(in the **Properties...** category) is the mixture critical temperature. This property is available when a composition dependent option is selected for critical temperature in the cases with Aungier-Redlich-Kwong real gas model and species transport

Current Magnitude

(in the **Potential...** category) is the magnitude of the electric current vector in the field, $\left|\sigma\nabla\varphi\right|$.

Curvature Correction Function fr

(in the **Turbulence...** category) is the multiplier of the production term when curvature correction is selected for Spalart-Allmaras or two equation models. See Curvature Correction for the Spalart-Allmaras and Two-Equation Models in the *Fluent Theory Guide* for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Custom Field Functions...
are scalar field functions defined by you. You can create a custom function using the Custom Field Function Calculator Dialog Box (p. 2683). All defined custom field functions will be listed in the lower drop-down list. See Custom Field Functions (p. 2003) for details.

Damkohler Number
(in the **Premixed Combustion...** category) is a nondimensional parameter that is defined as the ratio of turbulent to chemical time scales.

Density...
includes variables related to density.

Density
(in the **Density...** category) is the mass per unit volume of the fluid. Plots or reports of **Density** include only fluid cell zones. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list. The unit quantity for **Density** is **density**.

Density All
(in the **Density...** category) is the mass per unit volume of the fluid or solid material. Plots or reports of **Density All** include both fluid and solid cell zones. The unit quantity for **Density All** is **density**.

Derivatives...
are the viscous derivatives. For example, **dX-Velocity/dx** is the first derivative of the x component of velocity with respect to the x-coordinate direction. You can compute first derivatives of velocity, angular velocity, and pressure in the pressure-based solver, and first derivatives of velocity, angular velocity, temperature, and species in the density-based solvers.

DES Length Scale
(in the **Turbulence...** category) is available when you keep the temporary solver memory from being freed using the `solve/set/expert` text command. It is defined by Equation 4.250 in the Theory Guide for the Spalart-Allmaras based DES model and Equation 4.257 in the Theory Guide for the realizable k-epsilon based DES model. The DES length scale for the BSL / SST k-omega based DES model is defined by $C_{des}\Delta_{\max}=\left(\frac{L_t}{F_{DES}}\right)$, where C_{des} is a calibration constant used in the DES model and has a value of 0.61 and $\Delta_{\max}$ is the maximum local grid spacing ($\Delta x, \Delta y, \Delta z$).

DES TKE Dissipation Multiplier
(in the **Turbulence...** category) is either the multiplier function F_{DES} for Detached Eddy Simulation (DES) with BSL, SST, or Transition SST (see DES with the BSL or SST k-ω Model and DES with the Transition SST Model in the Theory Guide for details) or the function f_d for DES with Spalart-Allmaras or Realizable k-ε (see Theory Guide DES with the Spalart-Allmaras Model or DES with the Realizable k-ε Model, respectively).

Diameter
(in the **Properties...** category) is the diameter of particles, droplets, or bubbles of the secondary phase selected in the **Phase** drop-down list. Its unit quantity is **length**.

Diffusion Coef. of Scalar-n
(in the **User Defined Scalars...** category) is the diffusion coefficient for the n^{th} user-defined scalar transport equation. See the Fluent Customization Manual for details about defining user-defined scalars.

Discrete Phase Sources...
includes quantities related to the discrete phase model sources. See Modeling Discrete Phase (p. 1235) for details about this model.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discrete Phase Variables...
includes non-source quantities related to the discrete phase model. See Modeling Discrete Phase (p. 1235) for details about this model.

DPM Absorption Coefficient
(in the **Discrete Phase Variables...** category) is the absorption coefficient for discrete-phase calculations that involve radiation (a in Equation 5.17 in the Theory Guide). Its unit quantity is **length-inverse**.

DPM Accretion
(in the **Discrete Phase Variables...** category) is the accretion rate calculated at a wall boundary:

$$R_{accretion} = \sum_{p=1}^{N} \frac{\dot{m}_p}{A_{face}} \tag{33.8}$$

where $\dot{m}_p$ is the mass flow rate of the particle stream, and A_{face} is the area of the wall face where the particle strikes the boundary. This item will appear only if the optional erosion/accretion model is enabled. See Monitoring Erosion/Accretion of Particles at Walls (p. 1251) for details. The unit quantity for **DPM Accretion** is **mass-flux**.

DPM Burnout
(in the **Discrete Phase Sources...** category) is the exchange of mass from the discrete to the continuous phase for the combustion law (Law 5) and is proportional to the solid phase reaction rate. The burnout exchange has units of **mass-flow** and is reported as the rate occurring in each cell. A unit cell depth is used for 2D cases, and a reference cell depth of 1 radian is used for 2D axisymmetric cases.

DPM Collision Rate
(in the **Discrete Phase Variables...** category) is the time rate of particle-to-particle collisions per unit volume. Its unit quantity is **collision-rate**.

DPM Concentration
(in the **Discrete Phase Variables...** category) is the total concentration of the particles in all phases in a cell. Its unit quantity is **density**.

DPM Conc of <*component*>
(in the **Discrete Phase Variables...** category) is the mean concentration of <*component*> in the particles within a cell, computed per phase. Therefore, when using DDPM, this quantity appears for each combination of <*component*> and discrete particle phase (including the mixture phase). Its unit quantity is **density**.

DPM Density
(in the **Discrete Phase Variables...** category) is the mean particle density. This is computed as the mean density of the particles in each cell and is computed per phase. Therefore, when using DDPM this quantity is available for each discrete phase and the mixture phase. Its unit quantity is **density**.

DPM Diameter
(in the **Discrete Phase Variables...** category) is the mean particle diameter. This is computed as the mean diameter of the discrete phase particles in each cell and is computed per phase. Therefore, when using DDPM this quantity is available for each discrete phase and the mixture phase. Its unit quantity is **length**.

DPM Emission
(in the **Discrete Phase Variables...** category) is the amount of radiation emitted by a discrete-phase particle per unit volume. Its unit quantity is **heat-generation-rate**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DPM Enthalpy Source

(in the **Discrete Phase Sources...** category) is the exchange of enthalpy (sensible enthalpy plus heat of formation) from the discrete phase to the continuous phase. The exchange is positive when the particles are a source of heat in the continuous phase. The unit quantity for **DPM Enthalpy Source** is **power** and is reported as the rate of exchange occurring in each cell. A unit cell depth is used for 2D cases, and a reference cell depth of 1 radian is used for 2D axisymmetric cases.

DPM Erosion Rate (Generic)

(in the **Discrete Phase Variables...** category) is the erosion rate calculated at a wall boundary face using Equation 16.308 in the *Fluent Theory Guide* based on the values entered in the **Generic Erosion Model Parameters** dialog box (see Setting Particle Erosion and Accretion Parameters (p. 1309)). This item will appear only if the **Erosion/Accretion** option is selected in the **Physical Models** tab of the Discrete Phase Model Dialog Box (p. 2278) and the **Generic Model** option is enabled in the **DPM** tab of the Wall Dialog Box (p. 2456). See Monitoring Erosion/Accretion of Particles at Walls (p. 1251) for details. The unit quantity for **DPM Erosion Rate (Generic)** is **mass-flux**.

DPM Erosion Rate (Finnie)

(in the **Discrete Phase Variables...** category) is the Finnie erosion rate calculated using the Finnie formulation. (See Accretion in the *Fluent Theory Guide* for background information about this model.) This item will appear only if the **Finnie** option is enabled in the **Physical Models** tab of the Discrete Phase Model Dialog Box (p. 2278). See Monitoring Erosion/Accretion of Particles at Walls (p. 1251) for details. The unit quantity for **DPM Erosion Rate (Finnie)** is **mass-flux**.

DPM Erosion Rate (McLaury)

(in the **Discrete Phase Variables...** category) is the McLaury erosion rate calculated using the McLaury formulation. (See McLaury Erosion Model in the *Fluent Theory Guide* for background information about this model.) This item will appear only if the **McLaury** erosion model is enabled in the **Physical Models** tab of the Discrete Phase Model Dialog Box (p. 2278). See Monitoring Erosion/Accretion of Particles at Walls (p. 1251) for details. The unit quantity for **DPM Erosion Rate (McLaury)** is **mass-flux**.

DPM Erosion Rate (Oka)

(in the **Discrete Phase Variables...** category) is the Oka erosion rate calculated using the Oka formulation. (See Oka Erosion Model in the *Fluent Theory Guide* for background information about this model.) This item will appear only if the **Oka** erosion model is enabled in the **Physical Models** tab of the Discrete Phase Model Dialog Box (p. 2278). See Monitoring Erosion/Accretion of Particles at Walls (p. 1251) for details. The unit quantity for **DPM Erosion Rate (Oka)** is **mass-flux**.

DPM Evaporation/Devolatilization

(in the **Discrete Phase Sources...** category) is the exchange of mass, due to droplet-particle evaporation or combusting-particle devolatilization, from the discrete phase to the evaporating or devolatilizing species. If you are not using the non-premixed combustion model, the mass source for each individual species (**DPM species-n Source**, below) is also available; for non-premixed combustion, only this sum is available. The unit quantity for **DPM Evaporation/Devolatilization** is **mass-flow** and is reported as the rate of exchange occurring in each cell. A unit cell depth is used for 2D cases, and a reference cell depth of 1 radian is used for 2D axisymmetric cases.

DPM Granular Temperature

(in the **Discrete Phase Variables...** category) is the mean Granular Temperature for the discrete phase particles within a cell. Its unit quantity is **turbulent kinetic energy**. This quantity is only available when using a granular secondary phase in the Dense Discrete Phase Model.

DPM Mass Source

(in the **Discrete Phase Sources...** category) is the total exchange of mass from the discrete phase to the continuous phase. The mass exchange is positive when the particles are a source of mass in the continuous

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

phase. If you are not using the non-premixed combustion model, **DPM Mass Source** will be equal to the sum of all species mass sources (**DPM species-n Source**, below); if you are using the non-premixed combustion model, it will be equal to **DPM Burnout** plus **DPM Evaporation/Devolatilization**. The unit quantity for **DPM Mass Source** is **mass-flow** and is reported as the rate of exchange occurring in each cell. A unit cell depth is used for 2D cases, and a reference cell depth of 1 radian is used for 2D axisymmetric cases.

DPM D20

(in the **Discrete Phase Variables...** category) is the mean D20 diameter for the discrete phase particles within a cell. This is computed per phase. Therefore, when using DDPM this quantity is available for each discrete phase and the mixture phase. Its unit quantity is **length**. For the definition of D20 diameter, refer to Summary Reporting of Current Particles (p. 1358).

DPM D30

(in the **Discrete Phase Variables...** category) is the mean D30 diameter for the discrete phase particles within a cell. This is computed per phase. Therefore, when using DDPM this quantity is available for each discrete phase and the mixture phase. Its unit quantity is **length**. For the definition of D30 diameter, refer to Summary Reporting of Current Particles (p. 1358).

DPM D32

(in the **Discrete Phase Variables...** category) is the mean Sauter diameter for the discrete phase particles within a cell. This is computed per phase. Therefore, when using DDPM this quantity is available for each discrete phase and the mixture phase. Its unit quantity is **length**. For the definition of Sauter diameter, refer to Summary Reporting of Current Particles (p. 1358).

DPM Number Density

(in the **Discrete Phase Variables...** category) is the number of particles per unit cell volume. This is computed per phase. Therefore, when using DDPM this quantity is available for each discrete phase and the mixture phase. Its unit quantity is **volume-inverse**.

DPM Parcels in Cell

(in the **Discrete Phase Variables...** category) is the number of discrete phase parcels in each cell. This is computed per phase. Therefore, when using DDPM this quantity is available for each discrete phase and the mixture phase.

DPM Particles in Cell

(in the **Discrete Phase Variables...** category) is the number of discrete phase particles in each cell. This is computed per phase. Therefore, when using DDPM this quantity is available for each discrete phase and the mixture phase.

DPM RMS Diameter

(in the **Discrete Phase Variables...** category) is the RMS particle diameter. This is computed as the RMS particle diameter within each cell and is computed per phase. Therefore, when using DDPM, this quantity is available for each discrete phase and the mixture phase. Its unit quantity is length.

DPM RMS Temperature

(in the **Discrete Phase Variables...** category) is the RMS particle temperature. This is computed as the RMS particle temperature within each cell and is computed per phase. Therefore, when using DDPM this quantity is available for each discrete phase and the mixture phase. Its unit quantity is **temperature**.

DPM RMS X, Y, Z Velocity

(in the **Discrete Phase Variables...** category) are the RMS X, Y, and Z Velocity components of the discrete phase. These are computed as the mean discrete phase RMS velocities and are computed per phase.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Therefore, when using DDPM these quantities are available for each discrete phase and the mixture phase. Its unit quantity is **velocity**.

DPM Scattering

(in the **Discrete Phase Variables...** category) is the scattering coefficient for discrete-phase calculations that involve radiation (σ_s in Equation 5.17 in the Theory Guide). Its unit quantity is **length-inverse**.

DPM Sensible Enthalpy Source

(in the **Discrete Phase Sources...** category) is the exchange of sensible enthalpy from the discrete phase to the continuous phase. The exchange is positive when the particles are a source of heat in the continuous phase. Its unit quantity is **power** and is reported as the rate of exchange occurring in each cell. A unit cell depth is used for 2D cases, and a reference cell depth of 1 radian is used for 2D axisymmetric cases.

DPM species-n Source

(in the **Discrete Phase Sources...** category) is the exchange of mass, due to droplet-particle evaporation or combusting-particle devolatilization, from the discrete phase to the evaporating or devolatilizing species. (The name of the species will replace **species-n** in **DPM species-n Source**.) These species can be specified in the Set Injection Properties Dialog Box (p. 2796), and their descriptions can be found in Defining Injection Properties (p. 1286). The unit quantity is **mass-flow** and is reported as the rate of exchange occurring in each cell. A unit cell depth is used for 2D cases, and a reference cell depth of 1 radian is used for 2D axisymmetric cases. Note that this variable will not be available if you are using the non-premixed combustion model; use **DPM Evaporation/Devolatilization** instead.

DPM Specific Heat

(in the **Discrete Phase Variables...** category) is the mean particle specific heat. This is computed as the mean specific heat of the particles within each cell and is computed per phase. Therefore, when using DDPM this quantity is available for each discrete phase and the mixture phase. Its unit quantity is **specific-heat**.

DPM Swirl Momentum Source

(in the **Discrete Phase Sources...** category) is the exchange of swirl momentum from the discrete phase to the continuous phase. This value is positive when the particles are a source of momentum in the continuous phase. The unit quantity is **force** and is reported as the rate of exchange occurring in each cell. A unit cell depth is used for 2D cases, and a reference cell depth of 1 radian is used for 2D axisymmetric cases.

DPM Temperature

(in the **Discrete Phase Variables...** category) is the mean particle temperature. This is computed as the mean temperature of the particles within each cell and is computed per phase. Therefore, when using DDPM this quantity is available for each discrete phase and the mixture phase. Its unit quantity is **temperature**.

DPM Volume Fraction

(in the **Discrete Phase Variables...** category) is the volume fraction of the discrete phase. This is computed as the mean discrete phase volume fraction within each cell and is computed per phase. Therefore, when using DDPM this quantity is available for each discrete phase and the mixture phase.

DPM X, Y, Z Momentum Source

(in the **Discrete Phase Sources...** category) are the exchange of x-, y-, and z-direction momentum from the discrete phase to the continuous phase. These values are positive when the particles are a source of momentum in the continuous phase. The unit quantity is **force** and is reported as the rate of exchange occurring in each cell. A unit cell depth is used for 2D cases, and a reference cell depth of 1 radian is used for 2D axisymmetric cases.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DPM X, Y, Z Velocity

(in the **Discrete Phase Variables...** category) are the X, Y, and Z Velocity components of the discrete phase. These are computed as the mean discrete phase velocities and are computed per phase. Therefore, when using DDPM these quantities are available for each discrete phase and the mixture phase. Its unit quantity is **velocity**.

DRG Reduced Number of Reactions

(in the **Reactions…** category) is the number of retained reactions in reduced mechanism. This variable quantifies the size of the reduced mechanism at each cell or particle in the domain.

DRG Reduced Number of Species

(in the **Species...** category) is the number of retained species in reduced mechanism. This variable quantifies the size of the reduced mechanism at each cell or particle in the domain.

Dynamic Cell Volume Change

(in the **Mesh...** category) is the change of a cell volume.

Dynamic Pressure

(in the **Pressure...** category) is defined as $q \equiv \frac{1}{2}\rho v^2$. Its unit quantity is **pressure**.

Echem Reaction Rate of Reaction-n

(in the **Reactions...** category) is the n^{th} electrochemical reaction rate. This quantity is only available if the electrochemical reaction model is enabled.

Eff Diff Coef of species-n

(in the **Species...** category) is the sum of the laminar and turbulent diffusion coefficients of a species into the mixture:

$$D_{i,m} + \frac{\mu_t}{\rho Sc_t} \tag{33.9}$$

(The name of the species will replace **species-n** in **Eff Diff Coef of species-n**.) The unit quantity is **mass-diffusivity**.

Effective Prandtl Number

(in the **Turbulence...** category) is the ratio $\mu_{eff} c_p / k_{eff}$, where μ_{eff} is the effective viscosity, c_p is the specific heat, and k_{eff} is the effective thermal conductivity.

Effective Thermal Conductivity

(in the **Properties...** category) is the sum of the laminar and turbulent thermal conductivities, $k + k_t$, of the fluid. A large thermal conductivity is associated with a good heat conductor and a small thermal conductivity with a poor heat conductor (good insulator). Its unit quantity is **thermal-conductivity**.

Effective Viscosity

(in the **Turbulence...** category) is the sum of the laminar and turbulent viscosities of the fluid. Viscosity, μ, is defined by the ratio of shear stress to the rate of shear. Its unit quantity is **viscosity**.

Electric Potential

(in the **Potential...** category) is the electric potential φ used in Equation 7.92 in the *Fluent Theory Guide*.

Electric Conductivity

(in the **Properties...** category) is the electric conductivity σ used in Equation 7.80 in the *Fluent Theory Guide*.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Electrode Surface Potential

(in the **Potential...** category) is the electric potential on the electrode surface at an external Faradaic interface (φ_{ed} in the Butler-Volmer Equation 7.82 in the *Fluent Theory Guide*).

Enthalpy

(in the **Temperature...** category) is defined differently for compressible and incompressible flows, and depending on the solver and models in use.

For compressible flows,

$$H=\sum_j Y_j H_j \tag{33.10}$$

and for incompressible flows,

$$H=\sum_j Y_j H_j+\frac{p}{\rho} \tag{33.11}$$

where Y_j and H_j are, respectively, the mass fraction and enthalpy of species j. (See **Enthalpy of species-n**, below). For the pressure-based solver, the second term on the right-hand side of Equation 33.11 (p. 1970) is included only if the pressure work term is included in the energy equation (see Inclusion of Pressure Work and Kinetic Energy Terms in the Theory Guide). For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list. For all reacting flow models, the **Enthalpy** plots consist of the thermal (or sensible) plus chemical energy. The unit quantity for **Enthalpy** is **specific-energy**.

In the case of the inert model (Using the Non-Premixed Model with the Inert Model in the Theory Guide), the enthalpy in a cell is split into the contributions from the inert and the reacting fractions of the gas phase species in the cell. The cell enthalpy is partitioned as

$$H=\gamma H_{inert}+\left(1-\gamma\right)H_{pdf} \tag{33.12}$$

where γ is the fraction of inert species in the cell. The quantity H_{inert} is the enthalpy of the inert species at the cell temperature, similarly H_{pdf} is the enthalpy of the active species at the cell temperature. It is assumed that the cell temperature is common to both inert and active species, so H_{inert}, H_{pdf} and the cell temperature are chosen so that Equation 33.12 (p. 1970) is satisfied.

Enthalpy of species-n

(in the **Species...** category) is defined differently depending on the solver and models options in use. The quantity:

$$H_j=\int_{T_{ref,j}}^{T} c_{p,j}dT+h_j^0\left(T_{ref,j}\right) \tag{33.13}$$

where $h_j^0\left(T_{ref,j}\right)$ is the formation enthalpy of species j at the reference temperature $\left(T_{ref,j}\right)$, is reported only for non-adiabatic PDF cases, or if the density-based solver is selected. The quantity:

$$h_j=\int_{T_{ref}}^{T} c_{p,j}dT \tag{33.14}$$

where $T_{ref}=298.15K$, is reported in all other cases. The unit quantity for **Enthalpy of species-n** is **specific-energy**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entropy

(in the **Temperature...** category) is a thermodynamic property defined by the equation

$$\Delta S \equiv \int_{rev} \frac{\delta Q}{T} \tag{33.15}$$

$$\Delta S = C_p \ln\left(\frac{T}{T_{ref}}\right) - R \ln\left(\frac{P}{P_{ref}}\right) \tag{33.16}$$

the entropy is computed using the equation

$$\Delta S = C_p \ln\left(\frac{T}{T_{ref}}\right) \tag{33.17}$$

The unit quantity for entropy is **specific-heat**.

Important

Note that for the real gas models the entropy is computed accordingly by the appropriate equation of state formulation.

Existing Value

(in the **Adaption...** category) is the value that presently resides in the temporary space reserved for cell variables (that is, the last value that you displayed or computed).

Face Area Magnitude

(in the **Mesh...** category) is the magnitude of the face area vector for noninternal faces (that is, faces that only have `c0` and no `c1`). The values are stored on the face itself and used when required. This variable is intended only for zone surfaces and not for other surfaces created for postprocessing.

Face Handedness

(in the **Mesh...** category) is a parameter that is equal to one in cells that are adjacent to left-handed faces, and zero elsewhere. It can be used to locate mesh problems.

Faradaic Current Density

(in the **Potential...** category) is the electric current density produced by the electrochemical reaction at Faradaic interfaces (Equation 7.89 in the *Fluent Theory Guide*). Faradaic current density is positive if electric current flows from the electrode zone to the electrolyte zone, and negative if the electric current flows in the reverse direction.

Faradaic Heat Source

(in the **Potential...** category) is the energy source term due to the electrochemical reaction at Faradaic interfaces (Equation 7.94 in the *Fluent Theory Guide*).

Film Thickness

(in the **Eulerian Wall Film...** category) is the thickness of the wall film.

Film Mass

(in the **Eulerian Wall Film...** category) is the mass of the wall film.

Film Temperature

(in the **Eulerian Wall Film...** category) is the temperature of the wall film.

Film X-Velocity

(in the **Eulerian Wall Film...** category) is the x-component of the velocity of the wall film.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Film Y-Velocity
(in the **Eulerian Wall Film...** category) is the y-component of the velocity of the wall film.

Film Z-Velocity
(in the **Eulerian Wall Film...** category) is the z-component of the velocity of the wall film.

Film Velocity Magnitude
(in the **Eulerian Wall Film...** category) is the magnitude of the velocity of the wall film.

Film Effective Pressure
(in the **Eulerian Wall Film...** category) is the effective pressure of the wall film.

Film Surface X-Velocity
(in the **Eulerian Wall Film...** category) is the x-component of the surface velocity of the wall film.

Film Surface Y-Velocity
(in the **Eulerian Wall Film...** category) is the y-component of the surface velocity of the wall film.

Film Surface Z-Velocity
(in the **Eulerian Wall Film...** category) is the z-component of the surface velocity of the wall film.

Film Surface Velocity Magnitude
(in the **Eulerian Wall Film...** category) is the magnitude of the surface velocity of the wall film.

Film Surface Temperature
(in the **Eulerian Wall Film...** category) is the surface temperature of the wall film.

Film Passive Scalar
(in the **Eulerian Wall Film...** category) is the passive scalar for the wall film scalar equation.

Film Courant Number
(in the **Eulerian Wall Film...** category) is the Courant number of the wall film.

Film Weber Number
(in the **Eulerian Wall Film...** category) is the Weber number of the wall film.

Film Stripped Mass
(in the **Eulerian Wall Film...** category) is the cumulative mass of stripped wall film.

Film Stripped Diam
(in the **Eulerian Wall Film...** category) is the diameter of stripped wall film droplets.

Film DPM Mass Source
(in the **Eulerian Wall Film...** category) is the additional mass of discrete particles being absorbed into the wall film.

Film DPM Energy Source
(in the **Eulerian Wall Film...** category) is the additional energy of discrete particles being absorbed into the wall film.

Film DPM X-Momentum Source
(in the **Eulerian Wall Film...** category) is the x-component of any additional momentum of discrete particles being absorbed into the wall film.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Film DPM Y-Momentum Source

(in the **Eulerian Wall Film...** category) is the y-component of any additional momentum of discrete particles being absorbed into the wall film.

Film DPM Z-Momentum Source

(in the **Eulerian Wall Film...** category) is the z-component of any additional momentum of discrete particles being absorbed into the wall film.

Film X-Momentum Source

(in the **Eulerian Wall Film...** category) is the x-component of any additional momentum being absorbed into the wall film.

Film Y-Momentum Source

(in the **Eulerian Wall Film...** category) is the y-component of any additional momentum being absorbed into the wall film.

Fine Scale Mass Fraction of species-n

(in the **Species...** category) is the term Y_i^* in Equation 7.30 in the Theory Guide .

Fine Scale Temperature

(in the **Temperature...** category) is the temperature of the fine scales, which is calculated from the enthalpy when the reaction proceeds over the time scale (τ^* in Equation 7.29 in the Theory Guide), governed by the Kinetic rates of Equation 7.7 in the Theory Guide . Its unit quantity is **temperature**.

Fine Scale Transfer Rate

(in the **Species...** category) is the transfer rate of the fine scales, which is equal to the inverse of the time scale (τ^* in Equation 7.29 in the Theory Guide). Its unit quantity is **time-inverse**.

1-Fine Scale Volume Fraction

(in the **Species...** category) is a function of the fine scale volume fraction (ξ^* in Equation 7.28 in the Theory Guide). The quantity is subtracted from unity to make it easier to interpret.

Film Separated Mass

(in the **Eulerian Wall Film...** category) is the cumulative mass of separated wall film.

Fvar Prod

(in the **Pdf...** category) is the production term in the mixture fraction variance equation solved in the non-premixed combustion model (that is, the last two terms in Equation 8.5 in the Theory Guide).

Fvar2 Prod

(in the **Pdf...** category) is the production term in the secondary mixture fraction variance equation solved in the non-premixed combustion model. See Equation 8.5 in the Theory Guide .

Gas Constant (R)

(in the **Properties...** category) is the gas constant of the fluid. Its unit quantity is **specific-heat**.

Geometric Roughness Height

(in the **Turbulence...** category) is used in the roughness correlation of the Transition SST model. See Equation 4.187 in the Theory Guide.

Granular Conductivity

(in the **Properties...** category) is equivalent to the diffusion coefficient in Equation 17.321 in the Theory Guide . For more information, see Granular Temperature in the Theory Guide . Its unit quantity is kg/m-s.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Granular Pressure...

includes quantities for reporting the solids pressure for each granular phase (p_s in Equation 17.290 in the Theory Guide). See Solids Pressure in the Theory Guide for details. Its unit quantity is **pressure**. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Granular Temperature...

includes quantities for reporting the granular temperature for each granular phase (Θ_s in Equation 17.321 in the Theory Guide). See Granular Temperature in the Theory Guide for details. Its unit quantity is m^2/s^2. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

HCN Density

(in the **NOx...** category) is the mass per unit volume of HCN. The unit quantity is **density**. The **HCN Density** will appear only if you are modeling fuel NOx. See Fuel NOx Formation in the Theory Guide for details.

Heat of Heterogeneous Reaction

(in the **Phase Interaction...** category) is the heat added or removed due to heterogeneous chemical reactions. For exothermic reactions the **Heat of Heterogeneous Reaction** is reported as a positive quantity, while for endothermic reactions it will be a negative quantity. If you have more than one heterogeneous reaction defined in your case, the **Heat of Heterogeneous Reaction** reported is the sum of the heat for all heterogeneous reactions. The unit quantity of **Heat of Heterogeneous Reaction** is Watt.

Heat of Reaction

(in the **Reactions...** category) is the heat added or removed due to chemical reactions. The quantity reported by Fluent is the volumetric heat of reaction, as defined in Equation 5.10 in the Theory Guide multiplied by the cell volume. For exothermic reactions, the heat of reaction is reported as a positive quantity, while for endothermic reactions it is reported as a negative quantity. If you have more than one reaction defined in your case, the **Heat of Reaction** reported is the sum of the heat for all reactions. The unit of measurement for the heat of reaction is Watts. The **Heat of Reaction** is not available for the non-premixed and partially-premixed models.

Helicity

(in the **Velocity...** category) is defined by the dot product of vorticity and the velocity vector.

$$H=\left(\nabla\times\vec{V}\right)\cdot\vec{V} \tag{33.18}$$

Vorticity is a measure of the rotation of a fluid element as it moves in the flow field.

Incident Radiation

(in the **Radiation...** category) is the total radiation energy, G, that arrives at a location per unit time and per unit area:

$$G=\int_{\Omega=4\pi} I d\Omega \tag{33.19}$$

where I is the radiation intensity and Ω is the solid angle. G is the quantity that the P-1 radiation model computes. For the DO radiation model, the incident radiation is computed over a finite number of discrete solid angles, each associated with a vector direction. The unit quantity for **Incident Radiation** is **heat-flux**.

Incident Radiation (Band n)

(in the **Radiation...** category) is the radiation energy contained in the wavelength band $\Delta\lambda$ for the non-gray P-1 radiation model or the non-gray DO radiation model. Its unit quantity is **heat-flux**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Interface Overlap Fraction

(in the **mesh** category) is a metric for measuring the quality of the intersection between interface zones for non-conformal interfaces. It is calculated on each element of the interface zone and ranges from 0 to 1, where 0 means no intersection, and 1 means full intersection.

Intermittency

(in the **Turbulence...** category) is a measure of the probability that a given point is located inside a turbulent region. Upstream of transition the intermittency is zero. Once the transition occurs, the intermittency is ramped up to one until the fully turbulent boundary layer regime is achieved.

Intermittency Effective

(in the **Turbulence...** category) is used in the coupling of the Transition model and the SST Transport equations (see Coupling the Transition Model and SST Transport Equations in the Theory Guide for details).

Internal Energy

(in the **Temperature...** category) is the summation of the kinetic and potential energies of the molecules of the substance per unit volume (and excludes chemical and nuclear energies). **Internal Energy** is defined as $e=c_v T$. Its unit quantity is **specific-energy**.

Jet Acoustic Power

(in the **Acoustics...** category) is the acoustic power for turbulent axisymmetric jets (see Equation 15.15 in the Theory Guide). It is available only when the **Broadband Noise Sources** acoustics model is being used.

Jet Acoustic Power Level (dB)

(in the **Acoustics...** category) is the acoustic power for turbulent axisymmetric jets, reported in dB (see Equation 15.28 in the Theory Guide). It is available only when the **Broadband Noise Sources** acoustics model is being used.

Joule Heat Source

(in the **Potential...** category) is the energy source term due to the Joule heating (Equation 20.2 in the *Fluent Theory Guide*).

Kinetic Rate of Reaction-n

(in the **Reactions...** category) is given by the following expression (see Equation 7.7 in the Theory Guide for definitions of the variables shown here):

$$\hat{R}_r = \Gamma \left(k_{f,r} \prod_{j=1}^{N_r} [C_{j,r}]^{\eta'_{j,r}} - k_{b,r} \prod_{j=1}^{N_r} [C_{j,r}]^{\eta''_{j,r}} \right) \tag{33.20}$$

The reported value is independent of any particular species, and has units of kmol/m^3-s.

To find the rate of production/destruction for a given species i due to reaction r, multiply the reported reaction rate for reaction r by the term $M_i\left(\nu''_{i,r} - \nu'_{i,r}\right)$, where M_i is the molecular weight of species i, and $\nu''_{i,r}$ and $\nu'_{i,r}$ are the stoichiometric coefficients of species i in reaction r.

For particle reactions it is the global rate of the particle reaction n expressed in kmol/s/m3. This is computed as

$$\frac{\overline{R}_{j,r}}{M_j V}$$

where $\overline{R}_{j,r}$ is the rate of particle species depletion (or generation) given by Equation 7.69 in the Theory Guide , M_j is the particle species molecular weight, and V is the cell volume.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lam Diff Coef of species-n
(in the **Species...** category) is the laminar diffusion coefficient of a species into the mixture, $D_{i,m}$. Its unit quantity is **mass-diffusivity**.

Laminar Flame Speed
(in the **Premixed Combustion...** category) is the propagation speed of laminar premixed flames (U_l in Equation 9.9 in the Theory Guide). Its unit quantity is **velocity**.

Laminar Kinetic Energy (kl)
(in the **Turbulence...category**) is a measure of the "laminar" streamwise fluctuations present in the pre-transitional region of the boundary layer subjected to free-stream turbulence. A transport equation of kl is considered by the k-kl-omega transition model.

LEE Self-Noise X-Source, LEE Self-Noise Y-Source, LEE Self-Noise Z-Source
(in the **Acoustics...** category) are the self-noise source terms in the linearized Euler equation for the acoustic velocity component (see Equation 15.33 in the Theory Guide). They are available only when the **Broadband Noise Sources** acoustics model is being used.

LEE Shear-Noise X-Source, LEE Shear-Noise Y-Source, LEE Shear-Noise Z-Source
(in the **Acoustics...** category) are the shear-noise source terms in the linearized Euler equation for the acoustic velocity component (see Equation 15.33 in the Theory Guide). They are available only when the **Broadband Noise Sources** acoustics model is being used.

LEE Total Noise X-Source, LEE Total Noise Y-Source, LEE Total Noise Z-Source
(in the **Acoustics...** category) are the total noise source terms in the linearized Euler equation for the acoustic velocity component (see Equation 15.33 in the Theory Guide). The total noise source term is the sum of the self-noise and shear-noise source terms. They are available only when the **Broadband Noise Sources** acoustics model is being used.

LES Subgrid Turbulent Viscosity
(in the **Turbulence...** category) is the eddy viscosity that is determined by the local algebraic sub-grid scale model in an embedded LES zone, which actually affects the momentum transport equations (see Embedded Large Eddy Simulation (ELES) in the Theory Guide). Its unit quantity is **viscosity**.

Lilley's Self-Noise Source
(in the **Acoustics...** category) is the self-noise source term in the linearized Lilley's equation (see Equation 15.37 in the Theory Guide), available only when the **Broadband Noise Sources** acoustics model is being used.

Lilley's Shear-Noise Source
(in the **Acoustics...** category) is the shear-noise source term in the linearized Lilley's equation (see Equation 15.37 in the Theory Guide), available only when the **Broadband Noise Sources** acoustics model is being used.

Lilley's Total Noise Source
(in the **Acoustics...** category) is the total noise source term in the linearized Lilley's equation (see Equation 15.37 in the Theory Guide). The total noise source term is the sum of the self-noise and shear-noise source terms, available only when the **Broadband Noise Sources** acoustics model is being used.

Liquid Fraction
(in the **Solidification/Melting...** category) is the liquid fraction β computed by the solidification/melting model:

$$\beta = \frac{\Delta H}{L} = 0 \quad if \quad T < T_{solidus} \tag{33.21}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\beta = \frac{\Delta H}{L} = 1 \quad if \quad T > T_{liquidus} \tag{33.22}$$

$$\beta = \frac{\Delta H}{L} = \frac{T - T_{solidus}}{T_{liquidus} - T_{solidus}} \quad if \quad T_{solidus} < T < T_{liquidus} \tag{33.23}$$

log10(Shape Sensitivity Magnitude)

(in the **Sensitivities...** category) provides a convenience function that plots $\log_{10}$ of the magnitude, in view of the large range of values possible for the shape sensitivity magnitude. This allows the importance of the surfaces in a domain to be ranked more easily based on how they affect the observation of interest when they are reshaped.

Mach Number

(in the **Velocity...** category) is the ratio of velocity and speed of sound.

Magnitude of Sensitivity to Body Forces

(in the **Sensitivities...** category) is the magnitude of the adjoint velocity primitive field normalized by cell volume. This field can be interpreted as the magnitude of the sensitivity of the observable to body force per unit volume. It can be used to identify regions in the domain where small changes to the momentum balance in the flow can have a large or small effect on the observable. This field is often observed to be large, for example, upstream of a body for which drag sensitivity is of interest, with the field diminishing in the upstream direction. This indicates the interference effect for an object positioned at various locations upstream of the object of interest. This field is normalized so that, in principle, it shows a mesh-independent quantity. In practice, this field has features that are often correlated with mesh structure. This is not unexpected. In fact, the adjoint solution field is providing very useful insight into regions of the flow domain where the observation of interest is potentially sensitive to particular features of the mesh and discretization process that affect the local momentum balance.

Magnitude of Sensitivity to Body Forces (Cell Values)

(in the **Sensitivities...** category) is the magnitude of the adjoint velocity primitive field. This field can be interpreted at the cell level as the sensitivity of the observable to a body force on the cell, or any factors that may change the momentum balance on the cell. As such it is a mesh-dependent quantity. This field is often observed to be large, for example, upstream of a body for which drag sensitivity is of interest, with the field diminishing in the upstream direction. This indicates the interference effect for an object positioned at various locations upstream of the object of interest.

Mark Poor Elements

(in the **Mesh...** category) is a parameter that is equal to one in cells that are identified as invalid or poor, as well as cells that are adjacent to the face of an invalid or poor cell, and zero elsewhere. It can be used to mark and/or display invalid and poor elements.

Mass fraction of HCN, Mass fraction of NH3, Mass fraction of NO, Mass fraction of N2O

(in the **NOx...** category) are the mass of HCN, the mass of NH_3, the mass of NO, and the mass of N_2O per unit mass of the mixture (for example, kg of HCN in 1 kg of the mixture). The **Mass fraction of HCN** and the **Mass fraction of NH3** will appear only if you are modeling fuel NOx. See Fuel NOx Formation in the Theory Guide for details.

Mass fraction of nuclei

(in the **Soot...** category) is the number of particles per unit mass of the mixture (in units of particles $\times 10^{15}$/kg) The **Mass fraction of nuclei** will appear only if you use the two-step soot model. See Soot Formation (p. 1184) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mass fraction of soot

(in the **Soot...** category) is the mass of soot per unit mass of the mixture (for example, kg of soot in 1 kg of the mixture). See Soot Formation (p. 1184) for details.

Mass fraction of species-n

(in the **Species...** category) is the mass of a species per unit mass of the mixture (for example, kg of species in 1 kg of the mixture).

Mean *n*

(in the **Unsteady Statistics...** category) is the time-averaged value of a solution variable *n* (for example, **Static Pressure**). See Postprocessing for Time-Dependent Problems (p. 1622) for details.

Mean- *cff_n*

(in the **Unsteady Statistics...** category) is the time-averaged value of a custom field function *cff_n* (for example, **uns-custom-funtion-0**). See Postprocessing for Time-Dependent Problems (p. 1622) for details.

Mean DPM *n*

(in the **Unsteady DPM Statistics...** category) is the time-averaged value of a discrete phase variable *n* (for example, **Volume Fraction**). See Postprocessing for Time-Dependent Problems (p. 1622) for details.

Meridional Coordinate

(in the **Mesh...** category) is the normalized (dimensionless) coordinate that follows the flow path from inlet to outlet. Its value varies from 0 to 1.

Mesh...

includes variables related to the mesh.

Mesh X-Velocity, Mesh Y-Velocity, Mesh Z-Velocity

(in the **Velocity...** category) are the vector components of the mesh velocity for moving-mesh problems (rotating or multiple reference frames, mixing planes, or sliding meshes). Its unit quantity is **velocity**.

Mixture Fraction Variance

(in the **Pdf...** category) is the variance of the mixture fraction solved for in the non-premixed combustion model. This is the second conservation equation (along with the mixture fraction equation) that the non-premixed combustion model solves. (See Definition of the Mixture Fraction in the Theory Guide .)

Modified Turbulent Viscosity

(in the **Turbulence...** category) is the transported quantity $\tilde{\nu}$ that is solved for in the Spalart-Allmaras turbulence model and in DES with the Spalart-Allmaras model (see Equation 4.15 in the Theory Guide). The turbulent viscosity, μ_t, is computed directly from this quantity using the relationship given by Equation 4.16 in the Theory Guide . Its unit quantity is **viscosity**.

Molar Concentration of species-n

(in the **Species...** category) is the moles per unit volume of a species. Its unit quantity is **concentration**.

Mole fraction of species-n

(in the **Species...** category) is the number of moles of a species in one mole of the mixture.

Mole fraction of HCN, Mole fraction of NH3, Mole fraction of NO, Mole fraction of N2O

(in the **NOx...** category) are the number of moles of HCN, NH_3, NO, and N_2O in one mole of the mixture. The **Mole fraction of HCN** and the **Mole fraction of NH3** will appear only if you are modeling fuel NOx. See Fuel NOx Formation in the Theory Guide for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mole fraction of soot

(in the **Soot...** category) is the number of moles of soot in one mole of the mixture.

Molecular Prandtl Number

(in the **Properties...** category) is the ratio $c_p \mu_{lam} / k_{lam}$.

Molecular Viscosity

(in the **Properties...** category) is the laminar viscosity of the fluid. Viscosity, μ, is defined by the ratio of shear stress to the rate of shear. Its unit quantity is **viscosity**. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list. For granular phases, this is equivalent to the solids shear viscosity μ_s in Equation 17.307 in the Theory Guide .

Momentum Thickness Re ($Re_{\theta t}$)

(in the **Turbulence...** category) is based on the momentum thickness of the boundary layer. The SST transition model is considering a nonlocal empirical correlation for the value of $Re_{\theta t}$ in the free-stream, based on turbulence intensity, pressure gradient, and so on, and a transport equation to allow the free-stream value to diffuse into the boundary layer.

NH3 Density, NO Density, N2O Density

(in the **NOx...** category) are the mass per unit volume of NH_3, NO, and N_2O. The unit quantity for each is **density**. The **NH3 Density** will appear only if you are modeling fuel NOx. See Fuel NOx Formation in the Theory Guide for details.

Non-Equilibrium Thermal Model Source

(in the **Temperature...** category) is the scaled value of thermal conductivity for the fluid zone (γk_f) or for the overlapping solid zone ($(1-\gamma)k_s$), where γ is the porosity, k_f is the fluid phase thermal conductivity (including the turbulent contribution, k_t), and k_s is the solid medium thermal conductivity. These terms can be seen in Equation 6.14 (p. 266) and Equation 6.15 (p. 266).

See Non-Equilibrium Thermal Model Equations (p. 265) and Non-Equilibrium Thermal Model (p. 277) for details about the non-equilibrium thermal model.

Normal Optimal Displacement

(in the **Sensitivities...** category) shows the normal component of the optimal displacement computed from the adjoint solution. This field is defined only for portions of walls lying within the control-volume specified for morphing. The size of the optimal displacement is determined by the scale factor that is chosen for the morphing. A positive value of displacement indicates that the surface will be displaced into the flow domain, whereas a negative value of displacement corresponds to wall movement outwards from the flow domain. This field eliminates the component of the optimal displacement vector that lies in the plane of the wall.

Normal Shape Sensitivity

(in the **Sensitivities...** category) shows the normal component of the shape sensitivity. A positive value indicates an orientation directed into the domain, while a negative value indicates that the shape sensitivity is oriented outwards from the domain. This field eliminates the component of the vector shape sensitivity field that lies in the plane of the wall.

Normalized Concentration of Nuclei

(in the **Soot...** category) is the normalized soot particle number density.

Normalized Q criterion

(in the **Turbulence...** category) is a postprocessing quantity used for visual inspection of turbulence structures.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Normalized Soot Aggregation Moments
(in the **Soot...** category) is the normalized soot aggregate moments. This quantity is available only with the Method of Moments model.

Normalized soot moments
(in the **Soot...** category) are the normalized soot moments. This quantity is available only with the Method of Moments model.

NOx...
contains quantities related to the NOx model. See NOx Formation (p. 1153) for details about this model.

Orthogonal Quality
(in the **Mesh...** category) is a measure of the quality of a mesh, and is computed for each cell using the vectors from the cell centroid to each of its faces, the area vectors of the cell's faces, and the vectors from the cell centroid to the centroids of each of the adjacent cells (see Equation 5.1 (p. 131) and Equation 5.2 (p. 131)). The worst cells will have an **Orthogonal Quality** closer to 0, with better cells closer to 1.

Overset Cell Type
(in the **Cell Info...** category) is an integer value indicating the overset cell type. Each cell can have one of the following values:

Cell Type	Integer Function Value
donor	2
solve	1
receptor	0
orphan	–1
dead	–2

Overset Donor Count
(in the **Cell Info...** category) shows the number of donor cells a receptor cell has for data interpolation. A value of zero indicates that the cell is not a receptor, or that the receptor cell is an orphan cell.

Overset Receptor Count
(in the **Cell Info...** category) shows the number of receptor cells referencing a donor cell. A value of zero indicates that the cell is not a donor cell.

Primary Particle Diameter
(in the **Soot...** category) is a soot primary particle diameter within soot aggregates. This quantity is available only with the Method of Moments model.

Partition Boundary Cell Distance
(in the **Mesh...** category) is the smallest number of cells that must be traversed to reach the nearest partition (interface) boundary.

Partition Neighbors
(in the **Cell Info...** category) is the number of adjacent partitions (that is, those that share at least one partition boundary face (interface)). It gives a measure of the number of messages that will have to be generated for parallel processing.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pdf...

contains quantities related to the non-premixed combustion model, which is described in Modeling Non-Premixed Combustion (p. 1025).

PDF Table Adiabatic Enthalpy

is the adiabatic enthalpy corresponding to the cell value of mixture fraction. For single mixture fraction cases it is given by the following equation:

$$H_{ad}=H_{fuel}f+H_{oxidizer}\left(1-f\right) \tag{33.24}$$

and for cases involving a secondary stream it is given by the following equation:

$$H_{ad}=H_{fuel}f+H_{secondary}f_{sec}+H_{oxidizer}\left(1-f_{sec}-f\right) \tag{33.25}$$

where

f = mixture fraction

f_{sec} = secondary mixture fraction

H_{fuel} = total enthalpy of the fuel stream

$H_{secondary}$ = total enthalpy of the secondary stream

$H_{oxidizer}$ = total enthalpy of the oxidizer stream

For adiabatic cases the **PDF Table Adiabatic Enthalpy** is equal to the value of Enthalpy. The unit of measurement is specific-energy.

PDF Table Heat Loss/Gain

is given by the following equation:

$$h_{loss}=\left(H-H_{min}\right)/\left(H_{ad}-H_{min}\right)-1 \tag{33.26}$$

if the cell enthalpy is less than the adiabatic enthalpy, and by the following equation:

$$h_{gain}=1-\left(H_{max}-H\right)/\left(H_{max}-H_{ad}\right) \tag{33.27}$$

if the cell enthalpy is higher than adiabatic

where

H = total enthalpy

H_{ad} = the PDF Table Adiabatic Enthalpy

H_{min} = the minimum Enthalpy defined in the PDF table

H_{max} = the maximum Enthalpy defined in the PDF table

The **PDF Table Heat Loss/Gain** is dimensionless and ranges in value from -1, when H is equal to H_{min}, to +1, when H is equal to H_{max}. If H is equal to the adiabatic enthalpy it will be 0.

Phases...

contains quantities for reporting the volume fraction of each phase. See Modeling Multiphase Flows (p. 1365) for details.

Pitchwise Coordinate

(in the **Mesh...** category) is the normalized (dimensionless) coordinate in the circumferential (pitchwise) direction. Its value varies from 0 to 1.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Porous Deposition Rate of species-n
(in the **Species...** category) is the amount of a surface species that is deposited on porous zones. Its unit quantity is **mass-flux**.

Preconditioning Reference Velocity
(in the **Velocity...** category) is the reference velocity used in the coupled solver's preconditioning algorithm. See Preconditioning in the Theory Guide for details.

Premixed Combustion...
contains quantities related to the premixed combustion model, which is described in Modeling Premixed Combustion (p. 1089).

Pressure...
includes quantities related to a normal force per unit area (the impact of the gas molecules on the surfaces of a control volume).

Pressure Coefficient
(in the **Pressure...** category) is a dimensionless parameter defined by the equation

$$C_p = \frac{\left(p - p_{ref}\right)}{q_{ref}} \tag{33.28}$$

and q_{ref} is the reference dynamic pressure defined by $\frac{1}{2}\rho_{ref}v_{ref}{}^2$. The reference pressure, density, and velocity are defined in the Reference Values Task Page (p. 2502).

Pressure Spectrum Im *n*
(in the **Acoustics...** category) is the imaginary part of the complex Fourier amplitudes for Fourier mode *n*, as calculated by the Acoustic Sources FFT Dialog Box (p. 2549). See FFT of Acoustic Sources: Band Analysis and Export of Surface Pressure Spectra (p. 1222) for details.

Pressure Spectrum Re *n*
(in the **Acoustics...** category) is the real part of the complex Fourier amplitudes for Fourier mode *n*, as calculated by the Acoustic Sources FFT Dialog Box (p. 2549). See FFT of Acoustic Sources: Band Analysis and Export of Surface Pressure Spectra (p. 1222) for details.

Product Formation Rate
(in the **Premixed Combustion...** category) is the source term in the progress variable transport equation (S_c in Equation 9.1 in the Theory Guide). Its unit quantity is **time-inverse**.

Production of k
(in the **Turbulence...** category) is the rate of production of turbulence kinetic energy (times density). Its unit quantity is **turb-kinetic-energy-production**. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Production of laminar k
(in the **Turbulence...** category) is the production of laminar kinetic energy used in the k-kl-ω transition model. See Transport Equations for the k-kl-ω Model in the Theory Guide for more details (Equation 4.148).

Progress Variable
(in the **Premixed Combustion...** category) is a normalized mass fraction of the combustion products ($c=1$) or unburnt mixture products ($c=0$), as defined by Equation 9.2 in the Theory Guide .

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Properties...
includes material property quantities for fluids and solids.

Q criterion
(in the **Turbulence...** category) is a postprocessing quantity used for visual inspection of turbulence structures.

Rate of Coagulation
(in the **Soot...** category) is the rate of the coagulation of soot particles.

Rate of NO
(in the **NOx...** category) is the overall rate of formation of NO due to all active NO formation pathways (for example, thermal, prompt, and so on).

Rate of Nucleation
(in the **Soot...** category) is the rate of nucleation of the soot particles.

Rate of Nuclei
(in the **Soot...** category) is the overall rate of formation of nuclei.

Rate of N2OPath NO
(in the **NOx...** category) is the rate of formation of NO due to the N_2O pathway only (only available when N_2O pathway is active).

Rate of Oxidation
(in the **Soot...** category) is the rate of oxidation of the soot particles.

Rate of Prompt NO
(in the **NOx...** category) is the rate of formation of NO due to the prompt pathway only (only available when prompt pathway is active).

Rate of Reburn NO
(in the **NOx...** category) is the rate of formation of NO due to the reburn pathway only (only available when reburn pathway is active).

Rate of SNCR NO
(in the **NOx...** category) is the rate of formation of NO due to the SNCR pathway only (only available when SNCR pathway is active).

Rate of Soot
(in the **Soot...** category) is the overall rate of formation of soot mass.

Rate of Soot Mass Nucleation
(in the **Soot...** category) is the overall rate of soot nucleation.

Rate of Surface Growth
(in the **Soot...** category) is the rate of the soot particle surface growth due to reactions at soot particle surface.

Rate of Thermal NO
(in the **NOx...** category) is the rate of formation of NO due to the thermal pathway only (only available when thermal pathway is active).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rate of Fuel NO

(in the **NOx...** category) is the rate of formation of NO due to the fuel pathway only (only available when fuel pathway is active).

Rate of USER NO

(in the **NOx...** category) is the rate of formation of NO due to user defined rates only (only available when user-defined function rates are added).

Radial Coordinate

(in the **Mesh...** category) is the length of the radius vector in the polar coordinate system. The radius vector is defined by a line segment between the node and the axis of rotation. You can define the rotational axis in the Fluid Dialog Box (p. 2369). (See also Velocity Reporting Options (p. 1933).) The unit quantity for **Radial Coordinate** is **length**.

Radial Pull Velocity

(in the **Solidification/Melting...** category) is the radial-direction component of the pull velocity for the solid material in a continuous casting process. Its unit quantity is **velocity**.

Radial Velocity

(in the **Velocity...** category) is the component of velocity in the radial direction. (See Velocity Reporting Options (p. 1933) for details.) The unit quantity for **Radial Velocity** is **velocity**. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Radial-Wall Shear Stress

(in the **Wall Fluxes...** category) is the radial component of the force acting tangential to the surface due to friction. Its unit quantity is **pressure**.

Radiation...

includes quantities related to radiation heat transfer. See Modeling Radiation (p. 845) for details about the radiation models available in ANSYS Fluent.

Radiation Heat Flux

(in the **Wall Fluxes...** category) is the rate of radiation heat transfer through the control surface. It is calculated by the solver according to the specified radiation model. Heat flux out of the domain is negative, and heat flux into the domain is positive. The unit quantity for **Radiation Heat Flux** is **heat-flux**.

Radiation Intensity.Normalized Std Deviation

(in the **Radiation...** category) is the statistical standard deviation for the spherical mean radiation intensity.

Radiation Temperature

(in the **Radiation...** category) is the quantity θ_R, defined by

$$\theta_R = \left(\frac{G}{4\sigma}\right)^{1/4} \tag{33.29}$$

where G is the **Incident Radiation**. The unit quantity for **Radiation Temperature** is **temperature**.

Rate of Reaction-n

(in the **Reactions...** category) is the effective rate of progress of n^{th} reaction. For the finite-rate model, the value is the same as the **Kinetic Rate of Reaction-n**. For the eddy-dissipation model, the value is equivalent to the **Turbulent Rate of Reaction-n**. For the finite-rate/eddy-dissipation model, it is the lesser of the two.

For particle reactions it is the global rate of the particle reaction n expressed in kmol/s/m^3. This is computed as

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$\frac{\overline{R}_{j,r}}{M_j V}$

where $\overline{R}_{j,r}$ is the rate of particle species depletion (or generation) given by Equation 7.69 in the Theory Guide , M_j is the particle species molecular weight, and V is the cell volume.

Reactions...
includes quantities related to finite-rate reactions. See Modeling Species Transport and Finite-Rate Chemistry (p. 957) for information about modeling finite-rate reactions.

Reactor Net Mass Fraction of Species-n
(in the **Species...** category) is the mass of a species per unit mass of the mixture in the reactor network.

Reactor Net Temperature
(in the **Species...** category) is the computed temperature of the reactor network.

Reactor Net Zone ID
(in the **Species...** category) a unique identifier associated with each reactor network.

Reduced Temperature
(in the **Properties...** category) is the ratio T / T_c of the fluid temperature T divided by the critical temperature T_c. The reduced temperature T_r is available only with the Angier-Redlich-Kwong real gas model.

Reduced Pressure
(in the **Properties...** category) is the ratio P / P_c of the fluid pressure P divided by the critical pressure P_c. The reduced pressure P_r is available only with the Angier-Redlich-Kwong real gas model.

Reflected Radiation Flux (Band-n)
(in the **Wall Fluxes...** category) is the amount of radiative heat flux reflected by a semi-transparent wall for a particular band of radiation. Its unit quantity is **heat-flux**.

Reflected Visible Solar Flux, Reflected IR Solar Flux
(in the **Wall Fluxes...** category) is the amount of solar heat flux reflected by a semi-transparent wall or porous jump boundary for a visible or infrared (IR) radiation.

Refractive Index
(in the **Radiation...** category) is a nondimensional parameter defined as the ratio of the speed of light in a vacuum to that in a material. See Specular Semi-Transparent Walls in the Theory Guide for details.

Relative Axial Velocity
(in the **Velocity...** category) is the axial-direction component of the velocity relative to the reference frame motion. See Velocity Reporting Options (p. 1933) for details. The unit quantity for **Relative Axial Velocity** is **velocity**.

Relative Humidity
(in the **Species...** category) is the ratio of the partial pressure of the water vapor actually present in an air-water mixture to the saturation pressure of water vapor at the mixture temperature. ANSYS Fluent computes the saturation pressure, p, from the following equation [93] (p. 2892):

$$\ln\left(\frac{p}{p_c}\right) = \left(\frac{T_c}{T} - 1\right) \times \sum_{i=1}^{8} F_i \left[a\left(T - T_p\right) \right]^{i-1} \tag{33.30}$$

where

p_c = 22.089 MPa

T_c = 647.286 K

$F_1 = -7.4192420$

$F_2 = 2.9721000 \times 10^{-1}$

$F_3 = -1.1552860 \times 10^{-1}$

$F_4 = 8.6856350 \times 10^{-3}$

$F_5 = 1.0940980 \times 10^{-3}$

$F_6 = -4.3999300 \times 10^{-3}$

$F_7 = 2.5206580 \times 10^{-3}$

$F_8 = -5.2186840 \times 10^{-4}$

a = 0.01

T_p = 338.15 K

Relative Mach Number

(in the **Velocity...** category) is the nondimensional ratio of the relative velocity and speed of sound.

Relative Radial Velocity

(in the **Velocity...** category) is the radial-direction component of the velocity relative to the reference frame motion. (See Velocity Reporting Options (p. 1933) for details.) The unit quantity for **Relative Radial Velocity** is **velocity**.

Relative Swirl Velocity

(in the **Velocity...** category) is the tangential-direction component of the velocity relative to the reference frame motion, in an axisymmetric swirling flow. (See Velocity Reporting Options (p. 1933) for details.) The unit quantity for **Relative Swirl Velocity** is **velocity**.

Relative Tangential Velocity

(in the **Velocity...** category) is the tangential-direction component of the velocity relative to the reference frame motion. (See Velocity Reporting Options (p. 1933) for details.) The unit quantity for **Relative Tangential Velocity** is **velocity**.

Relative Total Pressure

(in the **Pressure...** category) is the stagnation pressure computed using relative velocities instead of absolute velocities; that is, for incompressible flows the dynamic pressure would be computed using the relative velocities. (See Velocity Reporting Options (p. 1933) for more information about relative velocities.) The unit quantity for **Relative Total Pressure** is **pressure**.

Relative Total Temperature

(in the **Temperature...** category) is the stagnation temperature computed using relative velocities instead of absolute velocities. (See Velocity Reporting Options (p. 1933) for more information about relative velocities.) The unit quantity for **Relative Total Temperature** is **temperature**.

Relative Velocity Angle

(in the **Velocity...** category) is similar to the **Velocity Angle** except that it uses the relative tangential velocity, and is defined as

$$\tan^{-1}\left(-\frac{\text{relative-tangential-velocity}}{\text{axial-velocity}}\right) \tag{33.31}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Its unit quantity is **angle**.

Relative Velocity Magnitude

(in the **Velocity...** category) is the magnitude of the relative velocity vector instead of the absolute velocity vector. The relative velocity ($\vec{w}$) is the difference between the absolute velocity ($\vec{v}$) and the mesh velocity. For simple rotation, the relative velocity is defined as

$$\vec{w} \equiv \vec{v} - \vec{\Omega} \times \vec{r} \tag{33.32}$$

where $\vec{\Omega}$ is the angular velocity of a moving reference frame about the origin and $\vec{r}$ is the position vector. (See Velocity Reporting Options (p. 1933) for details.) The unit quantity for **Relative Velocity Magnitude** is **velocity**.

Relative X Velocity, Relative Y Velocity, Relative Z Velocity

(in the **Velocity...** category) are the x-, y-, and z-direction components of the velocity relative to the reference frame motion. (See Velocity Reporting Options (p. 1933) for details.) The unit quantity for these variables is **velocity**.

Residuals...

contains different quantities for the pressure-based and density-based solvers:

In the density-based solvers, this category includes the corrections to the primitive variables pressure, velocity, temperature, and species, as well as the time rate of change of the corrections to these primitive variables for the current iteration (that is, residuals). Corrections are the changes in the variables between the current and previous iterations and residuals are computed by dividing a cell's correction by its physical time step. The total residual for each variable is the summation of the Euler, viscous, and dissipation contributions. The dissipation components are the vector components of the flux-like, face-based dissipation operator.

In the pressure-based solver, only the **Mass Imbalance** in each cell is reported (unless you have requested others, as described in Postprocessing Residual Values (p. 1632)). At convergence, this quantity should be small compared to the average mass flow rate.

RMS (species-n) Mass Fraction

(in the **Species...** category) is the root mean square value of the mass of a species per unit mass of the mixture.

RMSE *n*

(in the **Unsteady Statistics...** category) is the root mean squared error value of a solution variable *n* (for example, **Static Pressure**). See Postprocessing for Time-Dependent Problems (p. 1622) for details.

RMSE- *cff_n*

(in the **Unsteady Statistics...** category) is the root mean squared error value of a custom field function *cff_n* (for example, **uns-custom-function-0**). See Postprocessing for Time-Dependent Problems (p. 1622) for details.

RMS DPM *n*

(in the **Unsteady DPM Statistics...** category) is the root mean square value of a discrete phase variable *n* (for example, **Volume Fraction**). See Postprocessing for Time-Dependent Problems (p. 1622) for details.

Rothalpy

(in the **Temperature...** category) is defined as

$$I = h + \frac{w^2}{2} - \frac{u^2}{2} \tag{33.33}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where h is the enthalpy, w is the relative velocity magnitude, and u is the magnitude of the rotational velocity $\vec{u}=\vec{\omega}\times\vec{r}$.

Scalar-n

(in the **User Defined Scalars...** category) is the value of the n^{th} scalar quantity you have defined as a user-defined scalar. See the Fluent Customization Manual for more information about user-defined scalars.

Scalar Dissipation

(in the **Pdf...** category) is one of two parameters that describes the species mass fraction and temperature for a laminar flamelet in mixture fraction spaces. It is defined as

$$\chi=2D\left|\nabla f\right|^2 \tag{33.34}$$

where f is the mixture fraction and D is a representative diffusion coefficient (see The Flamelet Concept in the Theory Guide for details). Its unit quantity is **time-inverse**.

Scattering Coefficient

(in the **Radiation...** category) is the property of a medium that describes the amount of scattering of thermal radiation per unit path length for propagation in the medium. It can be interpreted as the inverse of the mean free path that a photon will travel before undergoing scattering (if the scattering coefficient does not vary along the path). The unit quantity for **Scattering Coefficient** is **length-inverse**.

Secondary Mean Mixture Fraction

(in the **Pdf...** category) is the mean ratio of the secondary stream mass fraction to the sum of the fuel, secondary stream, and oxidant mass fractions. It is the secondary-stream conserved scalar that is calculated by the non-premixed combustion model. See Definition of the Mixture Fraction in the Theory Guide .

Secondary Mixture Fraction Variance

(in the **Pdf...** category) is the variance of the secondary stream mixture fraction that is solved for in the non-premixed combustion model. See Definition of the Mixture Fraction in the Theory Guide .

Sensible Enthalpy

(in the **Temperature...** category) is available when any of the species models are active and displays only the thermal (sensible) enthalpy.

Sensitivity to Body Force X-Component, Sensitivity to Body Force Y-Component, and Sensitivity to Body Force Z-Component

(in the **Sensitivities...** category) are the components of the adjoint velocity primitive field normalized by cell volume. This field can be interpreted as the magnitude of the sensitivity of the observable to components of a body force per unit volume. Consider a body force distribution, expressed as a force per unit volume. The integral of the vector product of that distribution with the components of this field gives a first-order estimate of the net effect of the body force on the observation.

Sensitivity to Body Force X-Component (Cell Values), Sensitivity to Body Force Y-Component (Cell Values), and Sensitivity to Body Force Z-Component (Cell Values)

(in the **Sensitivities...** category) are the components of the adjoint velocity primitive field. These fields show the sensitivity of the observable to the various components of a body force that may be applied to a cell, or other factors that may change the momentum balance on the cell. As such, it is a mesh-dependent quantity.

Sensitivity to Boundary Heat Flux

(in the **Sensitivities...** category) is available when the adjoint energy equation is solved and is defined on walls where a heat flux boundary condition is imposed. The field shows the sensitivity of the observation

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of interest to variations in the boundary heat flux through the wall. Its properties are analogous to those of Sensitivity to Boundary Temperature.

Sensitivity to Boundary Pressure

(in the **Sensitivities...** category) is defined on boundaries where there is a user-specified pressure as part of a boundary condition, such as on a pressure outlet. The field shows the sensitivity of the observation of interest to variations in the boundary pressure across the flow boundary. It is interesting to note that even though the original boundary condition specification may be for a uniform pressure on the domain boundary, the effect of a non-uniform pressure perturbation is available, The effect of any specific boundary pressure change can be estimated as an integral of the product of the change to the pressure with the plotted sensitivity field. Viewing this field will also indicate whether or not the assumption of a uniform pressure is adequate for the simulation.

Sensitivity to Boundary Temperature

(in the **Sensitivities...** category) is available when the energy adjoint equation is solved and is defined on boundaries where a temperature boundary condition is applied. This includes walls, velocity inlets, mass-flow inlets, pressure inlets, and pressure outlets where a backflow temperature may be specified. The field shows the sensitivity of the observation of interest to variations in the boundary temperature across the boundaries. Note that even if the original boundary condition specification is for a uniform temperature on the boundary, the effect of a non-uniform temperature perturbation is available. The effect of any specific boundary temperature change can be estimated as an integral of the product of the change to the temperature with the plotted sensitivity field. This field can be used to indicate whether or not the assumption of a uniform temperature is adequate for the simulation.

Sensitivity to Boundary X-Velocity, Sensitivity to Boundary Y-Velocity, and Sensitivity to Boundary Z-Velocity

(in the **Sensitivities...** category) are defined on those boundaries where a user-specification of a boundary velocity is made for the original flow calculation. This includes no-slip walls. The field shows how sensitive the observable of interest is to changes in the boundary velocity at any point. It is interesting to note that even though the original boundary condition specification may be for a uniform velocity on the domain boundary, the effect of a non-uniform velocity perturbation is available, The effect of any specific boundary velocity change can be estimated as an integral of the vector product of the change to the velocity with the plotted sensitivity field. A plot of this quantity on a velocity inlet, for example, can be very useful for assessing whether or not the inlet is positioned too close to key parts of the system. That is, it addresses the question of whether or not the flow domain is too small to achieve a successful computation of the performance measure of interest. Viewing this field will also indicate whether or not the assumption of a uniform inflow is adequate.

Sensitivity to Energy Sources

(in the **Sensitivities...** category) is available when the energy adjoint is solved and is the primitive adjoint temperature field normalized by cell volume. It can be interpreted at the cell level as the sensitivity of the observable with respect to thermal energy sources or sinks in the domain. This field is normalized so that, in principle, it is a mesh-independent quantity.

Sensitivity to Energy Sources (Cell Values)

(in the **Sensitivities...** category) is available when the energy adjoint is solved and is the primitive adjoint temperature field. It can be interpreted at the cell level as the sensitivity of the observable with respect to changes in the thermal energy balance of the cell. As such, it is a mesh-dependent quantity.

Sensitivity to Flow Blockage

(in the **Sensitivities...** category) is provided as a convenient tool for identifying portions of the flow domain where the introduction of blockages or obstructions in the flow can affect the observation of interest. Consider a blockage in the flow that generates a reaction force on the flow that is proportional to the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

local flow speed, and acting in the opposite direction to the local flow: $F=-\alpha(r)v$ where α is a local coefficient for the reaction force. The local contribution of this force on the observation of interest is determined by the vector product of this force with the adjoint velocity field. The flow blockage field that is plotted is $-v\cdot\tilde{v}$, namely the negative of the vector product of the flow velocity and the adjoint velocity (Cell Value).

Sensitivity to Mass Sources

(in the **Sensitivities...** category) is the primitive adjoint pressure field normalized by cell volume. This field can be interpreted as the sensitivity of the observable with respect to mass sources or sinks in the domain. Consider a mass source/sink distribution, expressed as mass flow rate per unit volume. The integral of that distribution, weighted by the local value of this field, gives the effect of the sources/sinks on the observation. This field is normalized so that, in principle, it shows a mesh-independent quantity. In practice, this field has features that are often correlated with mesh structure. This is not unexpected. In fact the adjoint solution field is providing very useful insight into regions of the flow domain where the observation of interest is potentially sensitive to particular features of the mesh and discretization process that affect local mass balance. When plotted on a boundary, this field indicates the effect of the addition or removal of fluid from the domain upon the quantity of interest. It is important to note that in this scenario the effect of the momentum of the fluid that is added or removed is not taken into account. The boundary velocity sensitivity should be plotted if that effect is also of interest.

Sensitivity to Mass Sources (Cell Values)

(in the **Sensitivities...** category) provides a plot of the primitive adjoint pressure field. This field can be interpreted at the cell level as the sensitivity of the observable with respect to any changes to the mass balance on the cell. As such, it is a mesh-dependent quantity.

Sensitivity to Viscosity

(in the **Sensitivities...** category) shows the sensitivity of the quantity of interest to variations in the turbulent effective viscosity for a turbulent problem, or the laminar viscosity in a laminar case. The sensitivity is normalized by the cell volume to account for cell size variations in the mesh.

Shape Sensitivity Magnitude

(in the **Sensitivities...** category) is the magnitude of the sensitivity of the observable with respect to a deformation applied to the mesh (both boundary and interior mesh). When plotted on the surface of a body the locations where this quantity is large indicates where small changes to the surface shape can have a large effect on the observable of interest. If the shape sensitivity magnitude is small then the effect of shape changes in this region can be expected to have a small effect on the observable of interest. When viewing this field, it is often observed that the magnitude varies by many orders of magnitude. Contour plots will clearly draw attention to regions with the highest sensitivity (often sharp edges and corners). However, it should be remembered that a relatively small surface movement that is distributed over a large area can have a cumulative effect that is large.

Shape Sensitivity X-Component, Shape Sensitivity Y-Component, and Shape Sensitivity Z-Component

(in the **Sensitivities...** category) are the individual components of the sensitivity of the observable of interest with respect to the mesh node locations. It is plotted as cell data and is computed as the average of the nodal sensitivities for a given cell, divided by the cell volume. Note that for this discrete adjoint solver the sensitivity of the result with respect to node locations both on and off boundaries is computed. The normalization by cell volume indicates that the fields that are plotted are the weighting factors for a continuous spatial deformation field. (Note that the nodal sensitivity data itself is used when mesh morphing is performed, and predictions about the effect of shape changes are made.)

Shielding Function for SBES or SDES

(in the **Turbulence...** category) is the shielding function f_{SDES} used as part of the Stress-Blended Eddy Simulation turbulence model and the Shielded Detached Eddy Simulation turbulence model. For details,

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

see Stress-Blended Eddy Simulation (SBES) and Shielded Detached Eddy Simulation (SDES) in the *Fluent Theory Guide*, as well as Including the SDES or SBES Model with BSL, SST, and Transition SST Models (p. 805).

Skin Friction Coefficient

(in the **Wall Fluxes...** category) is a nondimensional parameter defined as the ratio of the wall shear stress and the reference dynamic pressure

$$C_f \equiv \frac{\tau_w}{\frac{1}{2}\rho_{ref} v_{ref}^2} \tag{33.35}$$

where τ_w is the wall shear stress, and ρ_{ref} and v_{ref} are the reference density and velocity defined in the Reference Values Task Page (p. 2502). For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Solar Heat Flux

(in the **Wall Fluxes...** category) is the rate of solar heat transfer through the control surface. Heat flux out of the domain is negative and heat flux into the domain is positive.

Solidification/Melting...

contains quantities related to solidification and melting.

Soot...

contains quantities related to the **Soot** model, which is described in Soot Formation (p. 1184).

Soot Density

(in the **Soot...** category) is the mass per unit volume of soot. The unit quantity is **density**. See Fuel NOx Formation in the Theory Guide for details.

Soot Mean Diameter

(in the **Soot...** category) is the average diameter of the soot particles. This quantity is available only with the Method of Moments model.

Soot Surface Area

(in the **Soot...** category) is the total surface area of the soot particles. This quantity is available only with the Method of Moments model.

Soot Volume fraction

(in the **Soot...** category) is the volume fraction of soot.

Sound Speed

(in the **Properties...** category) is the acoustic speed. It is computed from $\sqrt{\frac{\gamma p}{\rho}}$. Its unit quantity is **velocity**.

Important

Note that for the real gas models the sound speed is computed accordingly by the appropriate equation of state formulation.

Spanwise Coordinate

(in the **Mesh...** category) is the normalized (dimensionless) coordinate in the spanwise direction, from hub to casing. Its value varies from 0 to 1.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

species-n Source Term
(in the **Species...** category) is the source term in each of the species transport equations due to reactions. The unit quantity is always kg/m^3-s.

Species...
includes quantities related to species transport and reactions.

Specific Dissipation Rate (Omega)
(in the **Turbulence...** category) is the rate of dissipation of turbulence kinetic energy in unit volume and time. Its unit quantity is **time-inverse**.

Specific Heat (Cp)
(in the **Properties...** category) is the thermodynamic property of specific heat at constant pressure. It is defined as the rate of change of enthalpy with temperature while pressure is held constant. Its unit quantity is **specific-heat**.

Specific Heat Ratio (gamma)
(in the **Properties...** category) is the ratio of specific heat at constant pressure to the specific heat at constant volume.

Spinodal Temperature
(in the **Properties...** category) is the temperature of the gas phase at which the derivative of pressure with respect to molar volume becomes positive. The spinodal temperature defines the point beyond which the equation of state is no longer valid. If the temperature of your case approaches the spinodal temperature in some regions, this indicates that the flow conditions in these regions probably fall inside the saturation dome. The **Spinodal Temperature** is available only with the Cubic Equation of State Real Gas models.

SPL for Octave Band at *x* Hz (dB)
(in the **Acoustics...** category) is the surface pressure level (in decibels) of the standard technical octave band at the octave central frequency *x*, as calculated by the Acoustic Sources FFT Dialog Box (p. 2549). See FFT of Acoustic Sources: Band Analysis and Export of Surface Pressure Spectra (p. 1222) for details. Note that the transformation to the decibel units is done by default using the standard acoustic reference pressure value of 2 x 10^{-5} Pa; you can change this value using the Acoustics Model Dialog Box (p. 2289).

SPL for 1/3-Octave Band at *x* Hz (dB)
(in the **Acoustics...** category) is the surface pressure level (in decibels) of the standard technical 1/3-octave band at the 1/3-octave central frequency *x*, as calculated by the Acoustic Sources FFT Dialog Box (p. 2549). See FFT of Acoustic Sources: Band Analysis and Export of Surface Pressure Spectra (p. 1222) for details. Note that the transformation to the decibel units is done by default using the standard acoustic reference pressure value of 2 x 10^{-5} Pa; you can change this value using the Acoustics Model Dialog Box (p. 2289).

SPL for Const Width Band *n* (dB)
(in the **Acoustics...** category) is the surface pressure level (in decibels) of the user-defined constant band *n*, as calculated by the Acoustic Sources FFT Dialog Box (p. 2549). See FFT of Acoustic Sources: Band Analysis and Export of Surface Pressure Spectra (p. 1222) for details. Note that the transformation to the decibel units is done by default using the standard acoustic reference pressure value of 2 x 10^{-5} Pa; you can change this value using the Acoustics Model Dialog Box (p. 2289).

Static Pressure
(in the **Pressure...** category) is the static pressure of the fluid. It is a gauge pressure expressed relative to the prescribed operating pressure. The absolute pressure is the sum of the **Static Pressure** and the operating pressure. Its unit quantity is **pressure**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Static Temperature

(in the **Temperature...** and **Premixed Combustion...** categories) is the temperature in the fluid / solid. Its unit quantity is **temperature**. For a wall or shadow wall surface, the **Static Temperature** is the temperature of the adjacent fluid / solid cells; for a shell surface (see Postprocessing Shells (p. 843)), it is the temperature of the shell layer cells on the c0 side (that is, the side closer to the associated wall surface).

Note that **Static Temperature** will appear in the **Premixed Combustion...** category only for adiabatic premixed combustion calculations. See Postprocessing for Premixed Combustion Calculations (p. 1096).

Stored Cell Partition

(in the **Cell Info...** category) is an integer identifier designating the partition to which a particular cell belongs. In problems in which the mesh is divided into multiple partitions to be solved on multiple processors using the parallel version of ANSYS Fluent, the partition ID can be used to determine the extent of the various groups of cells. The active cell partition is used for the current calculation, while the stored cell partition (the last partition performed) is used when you save a case file. See Partitioning the Mesh Manually and Balancing the Load (p. 2033) for more information.

Strain Rate

(in the **Derivatives...** category) relates shear stress to the viscosity. Also called the shear rate ($\dot{\gamma}$ in Equation 7.35 (p. 492)), the strain rate is related to the second invariant of the rate-of-deformation tensor $\overline{\overline{D}}$. Its unit quantity is **time-inverse**. In 3D Cartesian coordinates, the strain rate, S, is defined as

$$\begin{aligned} S^2 = & \left[\frac{\partial u}{\partial x}\left(\frac{\partial u}{\partial x}+\frac{\partial u}{\partial x}\right)+\frac{\partial u}{\partial y}\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)+\frac{\partial u}{\partial z}\left(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}\right)\right]+ \\ & \left[\frac{\partial v}{\partial x}\left(\frac{\partial v}{\partial x}+\frac{\partial u}{\partial y}\right)+\frac{\partial v}{\partial y}\left(\frac{\partial v}{\partial y}+\frac{\partial v}{\partial y}\right)+\frac{\partial v}{\partial z}\left(\frac{\partial v}{\partial z}+\frac{\partial w}{\partial y}\right)\right]+ \\ & \left[\frac{\partial w}{\partial x}\left(\frac{\partial w}{\partial x}+\frac{\partial u}{\partial z}\right)+\frac{\partial w}{\partial y}\left(\frac{\partial w}{\partial y}+\frac{\partial v}{\partial z}\right)+\frac{\partial w}{\partial z}\left(\frac{\partial w}{\partial z}+\frac{\partial w}{\partial z}\right)\right] \end{aligned} \tag{33.36}$$

For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Stream Function

(in the **Velocity...** category) is formulated as a relation between the streamlines and the statement of conservation of mass. A streamline is a line that is tangent to the velocity vector of the flowing fluid. For a 2D planar flow, the stream function, ψ, is defined such that

$$\rho u \equiv \frac{\partial \psi}{\partial y} \rho v \equiv -\frac{\partial \psi}{\partial x} \tag{33.37}$$

constant values of stream function defining two streamlines is the mass rate of flow between the streamlines.

The accuracy of the stream function calculation is determined by the text command `/display/set/n-stream-func`.

Stretch Factor

(in the **Premixed Combustion...** category) is a nondimensional parameter that is defined as the probability of unquenched flamelets (G in Equation 9.15 in the Theory Guide).

Subcritical Condition

(in the **Properties...** category) has a value of 1 if the flow condition is subcritical and 0 if the flow condition is supercritical and is available with the Cubic Equation of State Real Gas models.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Subgrid Dissipation Rate

(in the **Turbulence...** category) is the turbulence dissipation rate of the unresolved eddies, eps_{sgs}, only active for the LES and Kinetic Energy Subgrid-Scale Model. It is defined as

$$eps_{sgs}=C_{eps}\cdot\frac{k_{sgs}^{3/2}}{L_s} \tag{33.38}$$

Its unit quantity is **turbulent-energy-diss-rate**.

Subgrid Dynamic Prandtl Number

(in the **Turbulence...** category) is used in the calculation of the subgrid-scale turbulent flux of a scalar φ, see Subgrid-Scale Models in the Theory Guide, Equation 4.282.

Subgrid Dynamic Sc of Species

(in the **Turbulence...** category) is used in the calculation of the subgrid-scale turbulent flux for Species (see Subgrid Dynamic Prandtl Number).

Subgrid Dynamic Viscosity Const

(in the **Turbulence...** category) is the Smagorinsky model constant as determined by the dynamic procedure described in Dynamic Smagorinsky-Lilly Model in the Theory Guide). Additional information with respect to the Embedded LES (E-LES) model can be found in Postprocessing for Turbulent Flows (p. 814).

Subgrid Filter Length

(in the **Turbulence...** category) is a mixing length for subgrid scales of the LES turbulence model (defined as L_S in Equation 4.285 in the Theory Guide).

Subgrid Kinetic Energy

(in the **Turbulence...** category) is the turbulence kinetic energy per unit mass of the unresolved eddies, k_{sgs}, calculated using a transport equation, only active for the LES and Kinetic Energy Subgrid-Scale Model. It is defined as

$$k_{sgs}=\frac{1}{2}\left(\overline{u_k^2}-\overline{u}_k^2\right) \tag{33.39}$$

Additional information with respect to the Embedded LES (E-LES) model can be found in Postprocessing for Turbulent Flows (p. 814). Its unit quantity is **turbulent-kinetic-energy**.

Subgrid Test–Filter Length

(in the **Turbulence...** category) is the test filter width $\hat{\Delta}$ described in Dynamic Smagorinsky-Lilly Model in the Theory Guide). Additional information with respect to the Embedded LES (E-LES) model can be found in Postprocessing for Turbulent Flows (p. 814).

Subgrid Turbulent Viscosity

(in the **Turbulence...** category) is the turbulent (dynamic) viscosity of the fluid calculated using the LES turbulence model. It expresses the proportionality between the anisotropic part of the subgrid-scale stress tensor and the rate-of-strain tensor. (See Equation 4.276 in the Theory Guide .) Its unit quantity is **viscosity**.

Subgrid Turbulent Viscosity Ratio

(in the **Turbulence...** category) is the ratio of the subgrid turbulent viscosity of the fluid to the laminar viscosity, calculated using the LES turbulence model.

Subtest Kinetic Energy

(in the **Turbulence...** category) is the turbulence kinetic energy of filtered eddies, kf, only active for the LES and Kinetic Energy Subgrid-Scale Model. It is defined as

$$k_f=L_1+L_2+L_3 \tag{33.40}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

with L_i being the normal components of the Leonard stress.

Its unit quantity is **turbulent-kinetic-energy**

Surface Acoustic Power
(in the **Acoustics...** category) is the **Acoustic Power** per unit area generated by boundary layer turbulence (see Equation 15.32 in the Theory Guide). It is available only when the **Broadband Noise Sources** acoustics model is being used. Its unit quantity is **power** per **area**.

Surface Acoustic Power Level (dB)
(in the **Acoustics...** category) is the **Acoustic Power** per unit area generated by boundary layer turbulence, and represented in dB (see Equation 15.32 in the Theory Guide). It is available only when the **Broadband Noise Sources** acoustics model is being used.

Surface Cluster ID
(in the **Radiation...** category) is used to view the distribution of surface clusters in the domain. Each cluster has a unique integer number (ID) associated with it.

Surface Corrosion Rate of species-n
(in the **Species...** category) is the rate of n^{th} solid species consumption/deposition. A positive value indicates corrosion of the solid species on the interface, and a negative value indicates deposition of the solid species.

Surface Coverage of species-n
(in the **Species...** category) is the amount of a surface species that is deposited on the substrate at a specific point in time.

Surface Deposition Rate of species-n
(in the **Species...** category) is the amount of a surface species that is deposited on the substrate. Its unit quantity is **mass-flux**.

Surface dpdt RMS, definition
(in the **Acoustics...** category) is the RMS value of the time-derivative of static pressure ($\partial p / \partial t$). It is available when the **Ffowcs-Williams & Hawkings** acoustics model is being used.

Surface Heat Transfer Coef.
(in the **Wall Fluxes...** category), as defined in ANSYS Fluent, is given by the equation

$$h_{eff} = \frac{q}{T_{wall} - T_{ref}} \tag{33.41}$$

T_{ref} is the reference temperature defined in the Reference Values Task Page (p. 2502). Note that T_{ref} is a constant value that should be representative of the problem. Its unit quantity is the **heat-transfer- coefficient**.

Surface Incident Radiation
(in the **Wall Fluxes...** category) is the net incoming radiation heat flux on a surface. Its unit quantity is **heat-flux**.

Surface Nusselt Number
(in the **Wall Fluxes...** category) is a local nondimensional coefficient of heat transfer defined by the equation

$$Nu = \frac{h_{eff} L_{ref}}{k} \tag{33.42}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface Stanton Number

(in the **Wall Fluxes...** category) is a nondimensional coefficient of heat transfer defined by the equation

$$St = \frac{h_{eff}}{\rho_{ref} v_{ref} c_p} \tag{33.43}$$

v_{ref} are reference values of density and velocity defined in the Reference Values Task Page (p. 2502), and c_p is the specific heat at constant pressure.

Swirl Pull Velocity

(in the **Solidification/Melting...** category) is the tangential-direction component of the pull velocity for the solid material in a continuous casting process. Its unit quantity is **velocity**.

Swirl Velocity

(in the **Velocity...** category) is the tangential-direction component of the velocity in an axisymmetric swirling flow. See Velocity Reporting Options (p. 1933) for details. The unit quantity for **Swirl Velocity** is **velocity**. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Swirl-Wall Shear Stress

(in the **Wall Fluxes...** category) is the swirl component of the force acting tangential to the surface due to friction. Its unit quantity is **pressure**.

Tangential Velocity

(in the **Velocity...** category) is the velocity component in the tangential direction. (See Velocity Reporting Options (p. 1933) for details.) The unit quantity for **Tangential Velocity** is **velocity**.

Temperature...

indicates the quantities associated with the thermodynamic temperature of a material.

Thermal Conductivity

(in the **Properties...** category) is a parameter (k) that defines the conduction rate through a material via Fourier's law ($q = -k\nabla T$). A large thermal conductivity is associated with a good heat conductor and a small thermal conductivity with a poor heat conductor (good insulator). Its unit quantity is **thermal-conductivity**. Note that when postprocessing solid material properties, the Thermal Conductivity will be displayed as zero if an anisotropic thermal conductivity model is being used for a solid material.

Thermal Diff Coef of species-n

(in the **Species...** category) is the thermal diffusion coefficient for the n^{th} species ($D_{T,i}$ in Equation 7.68 (p. 519), Equation 7.70 (p. 520), and Equation 7.74 (p. 521)). Its unit quantity is **viscosity**.

Time Step

(in the **Residuals...** category) is the local time step of the cell, Δt, at the current iteration level. Its unit quantity is **time**.

Time Step Scale

(in the **Species...** category) is the factor by which the time step is reduced for the stiff chemistry solver (available in the density-based solver only). The time step is scaled down based on an eigenvalue and positivity analysis.

Total Echem Heat Source

(in the **Potential...** category) is the summation of the Joule and Faradaic heat sources. The total heat source due to electrochemical chemical reactions is added to the energy equation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Total Energy

(in the **Temperature...** category) is the total energy per unit mass. Its unit quantity is **specific-energy**. For all species models, plots of **Total Energy** include the sensible, chemical and kinetic energies. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Total Enthalpy

(in the **Temperature...** category) is defined as $H+\frac{1}{2}v^2$ where H is the **Enthalpy**, as defined in Equation 5.7 in the Theory Guide , and v is the velocity magnitude. Its unit quantity is **specific-energy**. For all species models, plots of **Total Enthalpy** consist of the sensible, chemical and kinetic energies. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Total Enthalpy Deviation

(in the **Temperature...** category) is the difference between **Total Enthalpy** and the reference enthalpy, $H+\frac{1}{2}v^2-H_{ref}$, where H_{ref} is the reference enthalpy defined in the Reference Values Task Page (p. 2502). However, for non-premixed and partially premixed models, **Total Enthalpy Deviation** is the difference between **Total Enthalpy** and total adiabatic enthalpy (total enthalpy where no heat loss or gain occurs). The unit quantity for **Total Enthalpy Deviation** is **specific-energy**. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Total Pressure

(in the **Pressure...** category) is the pressure at the thermodynamic state that would exist if the fluid were brought to zero velocity and zero potential. For compressible flows, the total pressure is computed using isentropic relationships. For constant c_p, this reduces to:

$$p_0=p\left[1+\frac{\gamma-1}{2}M^2\right]^{\gamma/(\gamma-1)} \tag{33.44}$$

For incompressible flows (constant density fluid), we use Bernoulli's equation, $p_0=p+p_{dyn}$, where p_{dyn} is the local dynamic pressure. Its unit quantity is **pressure**.

Important

Note that in the postprocessing, the total pressure is presented as gauge pressure, for compressible and incompressible flows. If the total absolute pressure is needed, then add the value of the reference pressure to the total gauge pressure.

Total Surface Corrosion Rate

(in the **Potential...** category) is the total rate of solid species consumption/deposition. A positive value indicates corrosion of the solid species on the interface, and a negative value indicates deposition of the solid species. It's computed as the sum of the surface corrosion rates (**Surface Corrosion Rates of species-n** reported under the **Species** category) of all species.

Total Surface Heat Flux

(in the **Wall Fluxes...** category) is the rate of total heat transfer through the control surface. It is calculated by the solver according to the boundary conditions being applied at that surface. By definition, heat flux out of the domain is negative, and heat flux into the domain is positive. The unit quantity for **Total Surface Heat Flux** is **heat-flux**.

Total Temperature

(in the **Temperature...** category) is the temperature at the thermodynamic state that would exist if the fluid were brought to zero velocity. For compressible flows, the total temperature is computed from the total enthalpy using the current c_p method (specified in the Create/Edit Materials Dialog Box (p. 2302)). For

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

incompressible flows, the total temperature is equal to the static temperature, unless kinetic energy is explicitly added. The unit quantity for **Total Temperature** is **temperature**.

Transmitted Radiation Flux (Band-n)

(in the **Wall Fluxes...** category) is the amount of radiative heat flux transmitted by a semi-transparent wall for a particular band of radiation. Its unit quantity is **heat-flux**.

Transmitted Visible Solar Flux, Transmitted IR Solar Flux

(in the **Wall Fluxes...** category) is the amount of solar heat flux transmitted by a semi-transparent wall or porous jump boundary for a visible or infrared radiation.

Turbulence...

includes quantities related to turbulence. See Modeling Turbulence (p. 749) for information about the turbulence models available in ANSYS Fluent.

Turbulence Intensity

(in the **Turbulence...** category) is the ratio of the magnitude of the RMS turbulent fluctuations to the reference velocity:

$$I=\frac{\sqrt{\frac{2}{3}k}}{v_{ref}} \tag{33.45}$$

where k is the turbulence kinetic energy and v_{ref} is the reference velocity specified in the Reference Values Task Page (p. 2502). The reference value specified should be the mean velocity magnitude for the flow. Note that turbulence intensity can be defined in different ways, so you may want to use a custom field function for its definition. See Custom Field Functions (p. 2003) for more information.

Turbulent Dissipation Rate (Epsilon)

(in the **Turbulence...** category) is the turbulent dissipation rate. Its unit quantity is **turbulent-energy-diss-rate**. This quantity is available for the k-epsilon and k-omega based turbulence models, where the epsilon/omega relationship is defined as

$$\varepsilon=0.09k\omega \tag{33.46}$$

For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Turbulent Flame Speed

(in the **Premixed Combustion...** category) is the turbulent flame speed computed by ANSYS Fluent using Equation 9.9 in the Theory Guide . Its unit quantity is **velocity**.

Turbulent Kinetic Energy (k)

(in the **Turbulence...** category) is the turbulence kinetic energy per unit mass defined as

$$k=\frac{1}{2}\overline{u_i'u_i'} \tag{33.47}$$

Turbulent Rate of Reaction-n

(in the **Reactions...** category) is the rate of progress of the n^{th} reaction computed by Equation 7.25 or Equation 7.26 (in the Theory Guide). For the "eddy-dissipation" model, the value is the same as the **Rate of Reaction-n**. For the "finite-rate" model, the value is zero.

Turbulent Reynolds Number (Re_y)

(in the **Turbulence...** category) is a nondimensional quantity defined as

$$\frac{\rho d\sqrt{k}}{\mu_{lam}} \tag{33.48}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where k is turbulence kinetic energy, d is the distance to the nearest wall, and μ_{lam} is the laminar viscosity.

Turbulent Viscosity
(in the **Turbulence...** category) is the turbulent viscosity of the fluid computed using the turbulence model. Its unit quantity is **viscosity**. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list. Additional information with respect to the Embedded LES (E-LES) model can be found in Postprocessing for Turbulent Flows (p. 814).

Turbulent Viscosity (large-scale)
(in the **Turbulence...** category) is used in the k-kl-ω transition model.See Transport Equations for the k-kl-ω Model in the Theory Guide for more details (Equation 4.149).

Turbulent Viscosity (small-scale)
(in the **Turbulence...** category) is used in the k-kl-ω transition model.See Transport Equations for the k-kl-ω Model in the Theory Guide for more details (Equation 4.143).

Turbulent Viscosity Ratio
(in the **Turbulence...** category) is the ratio of turbulent viscosity to the laminar viscosity. Additional information with respect to the Embedded LES (E-LES) model can be found in Postprocessing for Turbulent Flows (p. 814).

udm-n
(in the **User Defined Memory...** category) is the value of the quantity in the n^{th} user-defined memory location.

Unburnt Fuel Mass Fraction
(in the **Premixed Combustion...** category) is the mass fraction of unburnt fuel. This function is available only for non-adiabatic models.

Unsteady Statistics...
includes mean and root mean square error (RMSE) values of solution variables and custom field functions derived from transient flow calculations.

User Defined Memory...
includes quantities that have been allocated to a user-defined memory location. See the separate Fluent Customization Manual for details about user-defined memory.

User-Defined Scalars...
includes quantities related to user-defined scalars. See the separate Fluent Customization Manual for information about using user-defined scalars.

UU Reynolds Stress
(in the **Turbulence...** category) is the $\overline{u'^2}$ stress.

UV Reynolds Stress
(in the **Turbulence...** category) is the $\overline{u'v'}$ stress.

UW Reynolds Stress
(in the **Turbulence...** category) is the $\overline{u'w'}$ stress.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Variance of Species

(in the **NOx...** category) is the variance of the mass fraction of a selected species in the flow field. It is calculated from Equation 14.113 in the Theory Guide .

Variance of Species 1, Variance of Species 2

(in the **NOx...** category) are the variances of the mass fractions of the selected species in the flow field. They are each calculated from Equation 14.113 in the Theory Guide .

Variance of Temperature

(in the **NOx...** category) is the variance of the normalized temperature in the flow field. It is calculated from Equation 14.113 in the Theory Guide.

Velocity...

includes the quantities associated with the rate of change in position with time. The instantaneous velocity of a particle is defined as the first derivative of the position vector with respect to time, $d\vec{r} / dt$, termed the velocity vector, $\vec{v}$.

Velocity Angle

(in the **Velocity...** category) is defined as follows:

For a 2D model,

$$\tan^{-1}\left(\frac{\text{y-velocity-component}}{\text{x-velocity-component}}\right) \tag{33.49}$$

For a 2D or axisymmetric model,

$$\tan^{-1}\left(\frac{\text{radial-velocity-component}}{\text{axial-velocity-component}}\right) \tag{33.50}$$

For a 3D model,

$$\tan^{-1}\left(\frac{\text{tangential-velocity-component}}{\text{axial-velocity-component}}\right) \tag{33.51}$$

Its unit quantity is **angle**.

Velocity Magnitude

(in the **Velocity...** category) is the speed of the fluid. Its unit quantity is **velocity**. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Volume fraction

(in the **Phases...** category) is the volume fraction of the selected phase in the **Phase** drop-down list.

Volumetric Absorbed Radiation

(in the **Radiation...** category) is the spherical mean absorbed radiation and, for a single spectral band, can be defined as:

$$4\pi I\alpha \tag{33.52}$$

where I is the radiation intensity and α is the absorption coefficient.

Volumetric Emitted Radiation

(in the **Radiation...** category) is the spherical mean emitted radiation and, for a single spectral band, can be defined as.

$$4\varepsilon\sigma T^4\alpha \tag{33.53}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where ε is the blackbody fraction, σ is the Stefan-Boltzmann constant, T is the temperature, and α is the absorption coefficient.

Vorticity Magnitude

(in the **Velocity...** category) is the magnitude of the vorticity vector. Vorticity is a measure of the rotation of a fluid element as it moves in the flow field, and is defined as the curl of the velocity vector:

$$\xi = \nabla \times \vec{V} \tag{33.54}$$

VV Reynolds Stress

(in the **Turbulence...** category) is the $\overline{v'^2}$ stress.

VW Reynolds Stress

(in the **Turbulence...** category) is the $\overline{v'w'}$ stress.

Wall Film Face Pressure

(in the **Wall Film…** category) is the face pressure of the wall film.

Wall Film Height

(in the **Wall Film…** category) is the height of the wall film.

Wall Film Heat Flux

(in the **Wall Film…** category) is the heat flux of the wall film to the wall.

Wall Film Impingement Mass Flux

(in the **Wall Film…** category) is the mass flux impinging on the wall film.

Wall Film Mass

(in the **Wall Film…** category) is the mass of the wall film (in units of mass).

Wall Film Temperature

(in the **Wall Film…** category) is the temperature of the wall film.

Wall Film Velocity Magnitude

(in the **Wall Film…** category) is the velocity magnitude of the wall film.

Wall Film X-Velocity

(in the **Wall Film…** category) is the x-component of the velocity of the wall film.

Wall Film Y-Velocity

(in the **Wall Film…** category) is the y-component of the velocity of the wall film.

Wall Film Z-Velocity

(in the **Wall Film…** category) is the z-component of the velocity of the wall film.

Wall Fluxes...

includes quantities related to forces and heat transfer at wall surfaces.

Wall Func. Heat Tran. Coef.

is generally defined by the equation

$$h_{eff} = \frac{\rho c_p u^*}{T_c^*} - \frac{D / \left(T_w - T_p \right)}{T_c^*} \tag{33.55}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where c_p is the specific heat, u^* is the near-wall turbulence velocity scale, D is the viscous heating (included only for variable density flows, Equation 4.321 in the *Fluent Theory Guide*), T_c^* is the dimensionless law-of-the-wall temperature defined for the standard wall functions in Equation 4.320 in the *Fluent Theory Guide*. For the enhanced wall treatment, the combined forms for D and T_c^* (Equation 4.320 and Equation 4.321 in the *Fluent Theory Guide*) are replaced by the corresponding functions obtained using the blending of Kader, explained in Enhanced Wall Treatment for Momentum and Energy Equations in the *Fluent Theory Guide*.

Wall Irradiation Flux.Normalized Std Deviation

(in the **Radiation...** category) is the statistical standard deviation for the hemispherical mean irradiation flux.

Wall Shear Stress

(in the **Wall Fluxes...** category) is the force acting tangential to the surface due to friction. Its unit quantity is **pressure**. For multiphase models, this value corresponds Equation 4.321 in the *Fluent Theory Guide* to the selected phase in the **Phase** drop-down list.

Wall Temperature

(in the **Temperature...** category) is the temperature on a surface (including wall surfaces, shadow wall surfaces, and shell surfaces). For an illustration of such surfaces for thin walls and shells, see Figure 6.43: A Thin Wall (p. 374) and Figure 12.5: A Boundary Wall with Shell Conduction (p. 839), respectively.

Wall Temperature (Thin)

(in the **Temperature...** category) is for thin walls only, and reports the temperature on the surface that is separated from the fluid / solid cells by the wall thickness, as shown in Figure 6.43: A Thin Wall (p. 374). Note that the wall thermal boundary conditions are applied on this surface.

Wall Yplus

(in the **Turbulence...** category) is a nondimensional parameter defined by the equation

$$y^+ = \frac{\rho u_\tau y_P}{\mu} \tag{33.56}$$

where $u_\tau = \sqrt{\tau_w / \rho_w}$ is the friction velocity, y_P is the distance from point P to the wall, ρ is the fluid density, and μ is the fluid viscosity at point P. See Near-Wall Treatments for Wall-Bounded Turbulent Flows in the Theory Guide for details. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Wall Ystar

(in the **Turbulence...** category) is a nondimensional parameter defined by the equation

$$y^* = \frac{\rho C_\mu^{1/4} k_P^{1/2} y_P}{\mu} \tag{33.57}$$

where k_P is the turbulence kinetic energy at point P, y_P is the distance from point P to the wall, ρ is the fluid density, and μ is the fluid viscosity at point P. See Near-Wall Treatments for Wall-Bounded Turbulent Flows in the Theory Guide for details.

WW Reynolds Stress

(in the **Turbulence...** category) is the $\overline{w'^2}$ stress.

X-Coordinate, Y-Coordinate, Z-Coordinate

(in the **Mesh...** category) are the Cartesian coordinates in the x-axis, y-axis, and z-axis directions respectively. The unit quantity for these variables is **length**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X Face Area, Y Face Area, Z Face Area
(in the **Mesh...** category) are the components of the face area vector for noninternal faces (that is, faces that only have `c0` and no `c1`). The values are stored on the face itself and used when required. These variables are intended only for zone surfaces and not for other surfaces created for postprocessing.

X Pull Velocity, Y Pull Velocity, Z Pull Velocity
(in the **Solidification/Melting...** category) are the x, y, and z components of the pull velocity for the solid material in a continuous casting process. The unit quantity for each is **velocity**.

X Velocity, Y Velocity, Z Velocity
(in the **Velocity...** category) are the components of the velocity vector in the x-axis, y-axis, and z-axis directions, respectively. The unit quantity for these variables is **velocity**. For multiphase models, these values correspond to the selected phase in the **Phase** drop-down list.

X-Vorticity, Y-Vorticity, Z-Vorticity
(in the **Velocity...** category) are the x, y, and z components of the vorticity vector.

X-Wall Shear Stress, Y-Wall Shear Stress, Z-Wall Shear Stress
(in the **Wall Fluxes...** category) are the x, y, and z components of the force acting tangential to the surface due to friction. The unit quantity for these variables is **pressure**. For multiphase models, these values correspond to the selected phase in the **Phase** drop-down list.

33.5. Custom Field Functions

In addition to the basic field variables provided by ANSYS Fluent (and described in Alphabetical Listing of Field Variables and Their Definitions (p. 1957)), you can also define your own field functions to be used in conjunction with any of the commands that use these variables (contour and vector display, XY plots, and so on). This capability is available with the Custom Field Function Calculator Dialog Box (p. 2683). You can use the default field variables, previously defined calculator functions, and calculator operators to create new functions. (Several sample functions are described in Sample Custom Field Functions (p. 2007).)

Any field functions that you define will be saved in the case file the next time that you save it. You can also save your custom field functions to a separate file (as described in Manipulating, Saving, and Loading Custom Field Functions (p. 2006)), so that they can be used with a different case file.

Important

Note that all custom field functions are evaluated and stored in SI units.

Any solver-defined flow variables that you use in your field-function definition will be automatically converted if they are not already in SI units, but you must be careful to enter constants in the appropriate units. Note also that explicit node values are not available for custom field functions; all node values for these functions will be computed by averaging the values in the surrounding cells, as described in Node Values (p. 1931).

For additional information, see the following sections:

33.5.1. Creating a Custom Field Function
33.5.2. Manipulating, Saving, and Loading Custom Field Functions
33.5.3. Sample Custom Field Functions

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

33.5.1. Creating a Custom Field Function

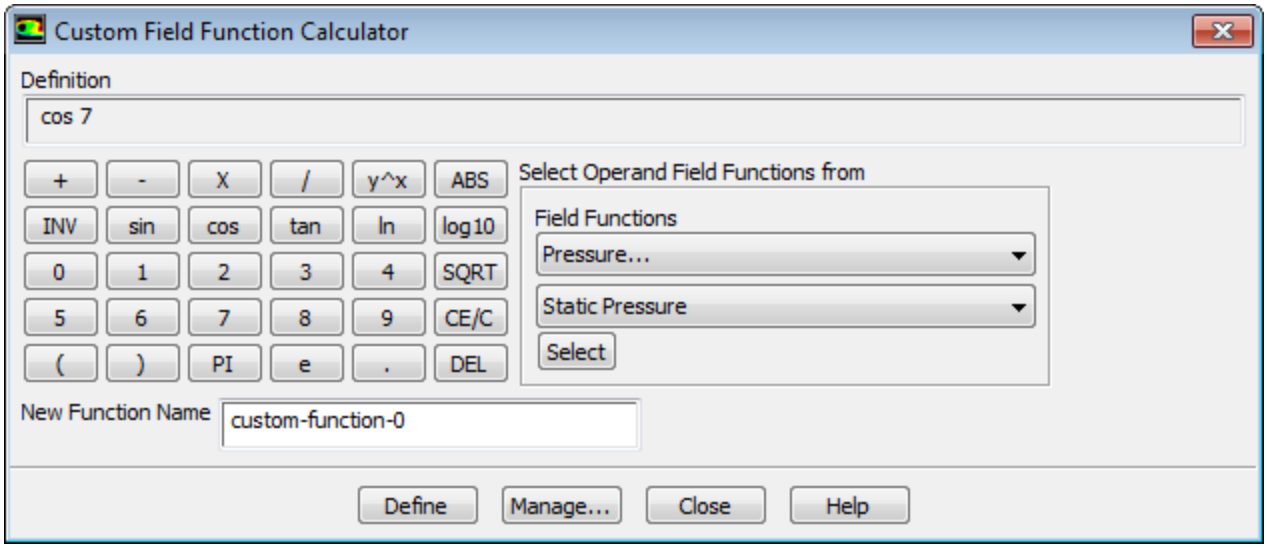
To create your own field function, you will use the Custom Field Function Calculator Dialog Box (p. 2683) (Figure 33.3: The Custom Field Function Calculator Dialog Box (p. 2004)). This dialog box allows you to define field functions based on existing functions, using simple calculator operators. Any functions that you define will be added to the list of default flow variables and other field functions provided by the solver.

Parameters & Customization → Custom Field Functions New...

Important

Recall that you must enter all constants in the function definition in SI units.

Figure 33.3: The Custom Field Function Calculator Dialog Box



The steps for creating a custom field function are as follows:

1. Use the calculator buttons and the **Field Functions** list and **Select** button to specify the function definition, as described below. (As you select each item from the **Field Functions** list or click a button in the calculator keypad, its symbol will appear in the **Definition** text entry box. You *cannot* edit the contents of this box directly; if you want to delete part of a function, use the **DEL** button on the keypad.)

   Important

   The range of integers and real numbers that can be stored is as follows:

   $-2147483648 > integers < 2147483647$

   $-1.79769e{+}308 > real < 1.79769e{+}308$

   Note that using a number less than 1e-39 may produce inaccurate results, while values less than 1e-45 will produce a result of zero.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Specify the name of the function in the **New Function Name** field.

> **Important**
>
> Be sure that you do not specify a name that is already used for a standard field function (for example, `velocity-magnitude`); you can see a complete list of the predefined field functions in ANSYS Fluent by selecting the `display/contours` text command and viewing the available choices for `contours of`.

3. Click the **Define** button.

When you click **Define**, the solver will create the function and add it to the list of **Custom Field Functions** within the drop-down list of available field functions. The **Define** push button is grayed out after you create a new function or if the **Definition** text entry box is empty.

Should you decide to rename or delete the function after you have completed the definition, you can do so in the Field Function Definitions Dialog Box (p. 2704), which you can open by clicking on the **Manage...** push button. See Manipulating, Saving, and Loading Custom Field Functions (p. 2006) for details.

33.5.1.1. Using the Calculator Buttons

Your function definition can include many basic calculator operations (for example, addition, subtraction, multiplication, square root). When you select a calculator button (by clicking on it), the appropriate symbol will appear in the **Definition** text entry box. The meaning of the buttons is straightforward; they are similar to the buttons you would find on any standard calculator. You should, however, note the following:

- The **CE/C** button will clear the entire **Definition** and the **New Function Name**, if you have entered one. The **DEL** button will delete only the last entry in the **Definition** text entry box. You can use **DEL** to delete characters one at a time, starting with the last one entered.
- To obtain the inverse trigonometric functions arcsin, arccos, and arctan, click the **INV** button before selecting **sin**, **cos**, or **tan**.
- The **ABS** button yields the absolute value of the number that follows it. Likewise, the **ln** button yields the natural logarithm of the number that follows it, and the **log10** button yields the base 10 logarithm function of the number that follows it.

> **Important**
>
> **log10** and **ln** will be calculated for values greater than 0. For values less than or equal to 0, the resultant value will be zero.

- The **PI** button represents π and the **e** button represents the base of the natural logarithm system (which is approximately equal to 2.71828).

33.5.1.2. Using the Field Functions List

Your function definition can include any of the field functions defined by the solver (and listed in Alphabetical Listing of Field Variables and Their Definitions (p. 1957)) or by you. You can also use any report

definitions that you have created (see Monitoring and Reporting Solution Data (p. 1885) for additional information). To include one of these variables/functions in your function definition, select it in the **Field Functions** drop-down list and then click the **Select** button below the list. The symbol for the selected item will appear in the **Definition** text entry box (for example, **p** will appear if you select **Static Pressure**).

Note

If a custom field function definition uses a report definition, then it cannot be exported to a file.

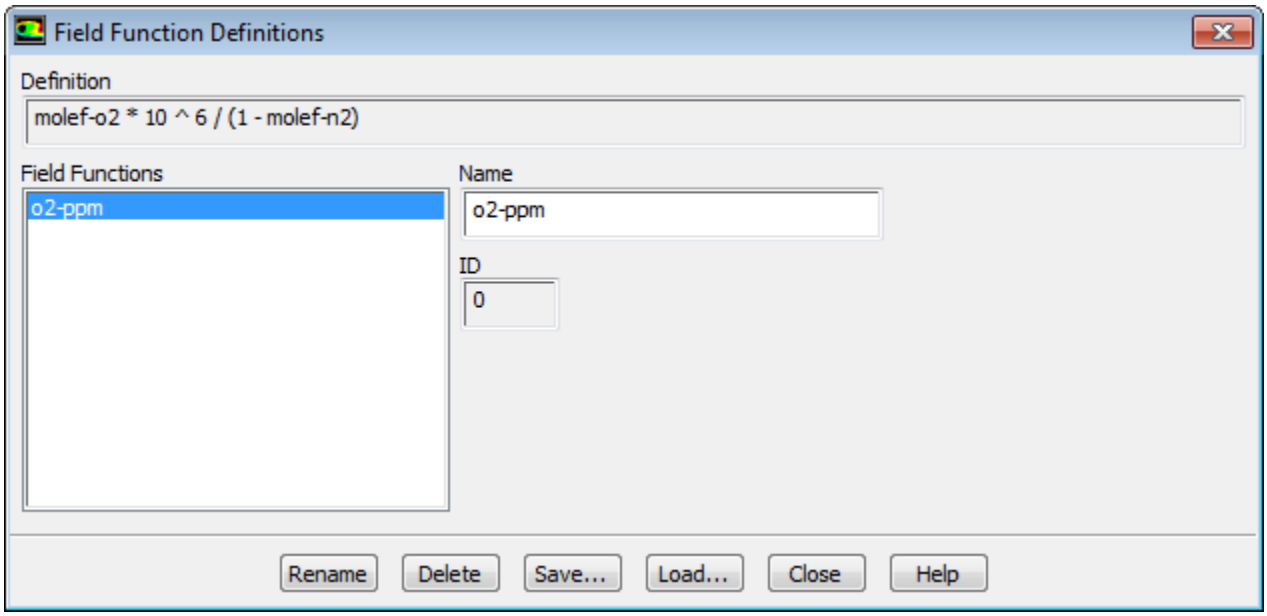
33.5.2. Manipulating, Saving, and Loading Custom Field Functions

Once you have defined your field functions, you can manipulate them using the Field Function Definitions Dialog Box (p. 2704) (Figure 33.4: The Field Function Definitions Dialog Box (p. 2006)). You can display a function definition to be sure that it is correct, delete the function if you decide that it is incorrect and must be redefined, or give the function a new name. You can also save custom field functions to a file or read them from a file. The custom field function file allows you to transfer your custom functions between case files.

To open the Field Function Definitions Dialog Box (p. 2704), right-click **Custom Field Functions** (under the **Parameters & Customization** branch of the tree) and select **Manage...**. You can also click the **Manage...** button in the Custom Field Function Calculator Dialog Box (p. 2683).

To open the Field Function Definitions Dialog Box (p. 2704), right-click **Custom Field Functions** (under the **Parameters & Customization** branch of the tree) and select **Manage...**. You can also click the **Manage...** button in the Custom Field Function Calculator Dialog Box (p. 2683).

Figure 33.4: The Field Function Definitions Dialog Box



The following actions can be performed in the Field Function Definitions Dialog Box (p. 2704):

- To check the definition of a function, select it in the **Field Functions** list. Its definition will be displayed in the **Definition** field. This display is for informational purposes only; you cannot edit it. If you want to change

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a function definition, you must delete the function and define it again in the Custom Field Function Calculator Dialog Box (p. 2683).

- To delete a function, select it in the **Field Functions** list and click the **Delete** button.
- To rename a function, select it in the **Field Functions** list, enter a new name in the **Name** field, and click the **Rename** button.

Important

Be sure that you do not specify a name that is already used for a standard field function (for example, `velocity-magnitude`); you can see a complete list of the predefined field functions in ANSYS Fluent by selecting the `display/contours` text command and viewing the available choices for `contours of`.

- To save all of the functions in the **Field Functions** list to a file, click the **Save...** button and specify the file name in The Select File Dialog Box (p. 19).
- To read custom field functions from a file that you saved as described above, click the **Load...** button and specify the file name in the resulting **Select File** dialog box. (Custom field function files are valid Scheme functions, and can also be loaded with the **File/Read/Scheme...** ribbon tab item, as described in Reading Scheme Source Files (p. 57).)

33.5.3. Sample Custom Field Functions

When you are checking the results of your simulation, you may find it useful to define some of the following field functions:

- To define a function that determines the ratio of static pressure to inlet total pressure, use the relationship

$$R=\frac{p+p_{op}}{p_{to}+p_{op}} \tag{33.58}$$

where p is the static pressure calculated by the solver, p_{to} is the inlet total pressure, and p_{op} is the operating pressure for the problem. Use the solver-defined function **Static Pressure** for p, and the numerical value that you specified for **Gauge Total Pressure** in the Pressure Inlet Dialog Box (p. 2434) for p_{to}. Specify the value of the operating pressure to be the value that you set in the Operating Conditions Dialog Box (p. 2382). As discussed in Operating Pressure (p. 530), all pressures in ANSYS Fluent are gauge pressures relative to the operating pressure. If the operating pressure is zero, as is generally the case for compressible flow calculations, the expression for the pressure ratio reduces to

$$PR=\frac{p}{p_{to}} \tag{33.59}$$

- To define a function that determines the critical velocity ratio v/a_*, a parameter that is sometimes used in turbomachinery calculations, use the relationship

$$\frac{v}{a_*}=\left[\left(\frac{\gamma+1}{\gamma-1}\right)\left(1-PR^{(\gamma-1)/\gamma}\right)\right]^{1/2} \tag{33.60}$$

In this relationship, a_* is the critical velocity (that is, the velocity that would occur for the same stagnation conditions if $M=1$), γ is the ratio of specific heats, and PR is the pressure ratio defined in

Equation 33.59 (p. 2007) for which you created your own function. For γ, ratio of specific heats, select **Specific Heat Ratio (gamma)** in the **Properties...** category. To include PR, select **Custom Field Functions...** in the first drop-down list under **Field Functions**, and then select from the second list the function name that you assigned PR.

- Suppose you have swirling flow in a pipe, aligned with the z axis, and you want to calculate the flow rate of angular momentum through a cross-sectional plane:

$$\int \rho r v_\theta \vec{v} \cdot d\vec{A} \tag{33.61}$$

You can create a function for the product rv_θ, where r is the **Radial Coordinate** and v_θ is the **Tangential Velocity**. Then use the Surface Integrals Dialog Box (p. 2616) to compute the flow rate of this quantity.

Important

The custom field function containing model dependent functions (like temperature when the energy equation is enabled) will be computed only when those models are still active.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 34: Parallel Processing

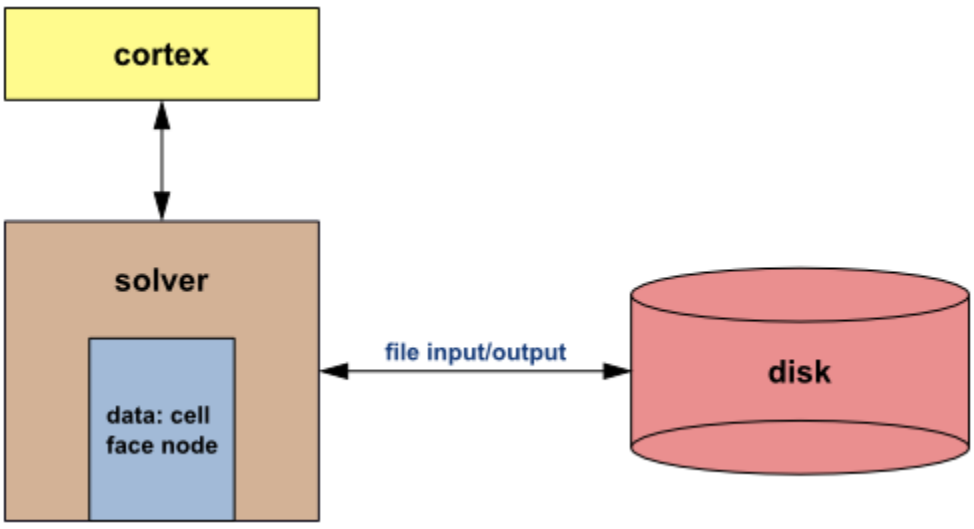
The following sections describe the parallel-processing features of ANSYS Fluent.

34.1. Introduction to Parallel Processing
34.2. Starting Parallel ANSYS Fluent Using Fluent Launcher
34.3. Starting Parallel ANSYS Fluent on a Windows System
34.4. Starting Parallel ANSYS Fluent on a Linux System
34.5. Mesh Partitioning and Load Balancing
34.6. Using General Purpose Graphics Processing Units (GPGPUs) With the Algebraic Multigrid (AMG) Solver
34.7. Controlling the Threads
34.8. Checking Network Connectivity
34.9. Checking and Improving Parallel Performance

34.1. Introduction to Parallel Processing

The ANSYS Fluent serial solver manages file input and output, data storage, and flow field calculations using a single solver process on a single computer (Figure 34.1: Serial ANSYS Fluent Architecture (p. 2009)).

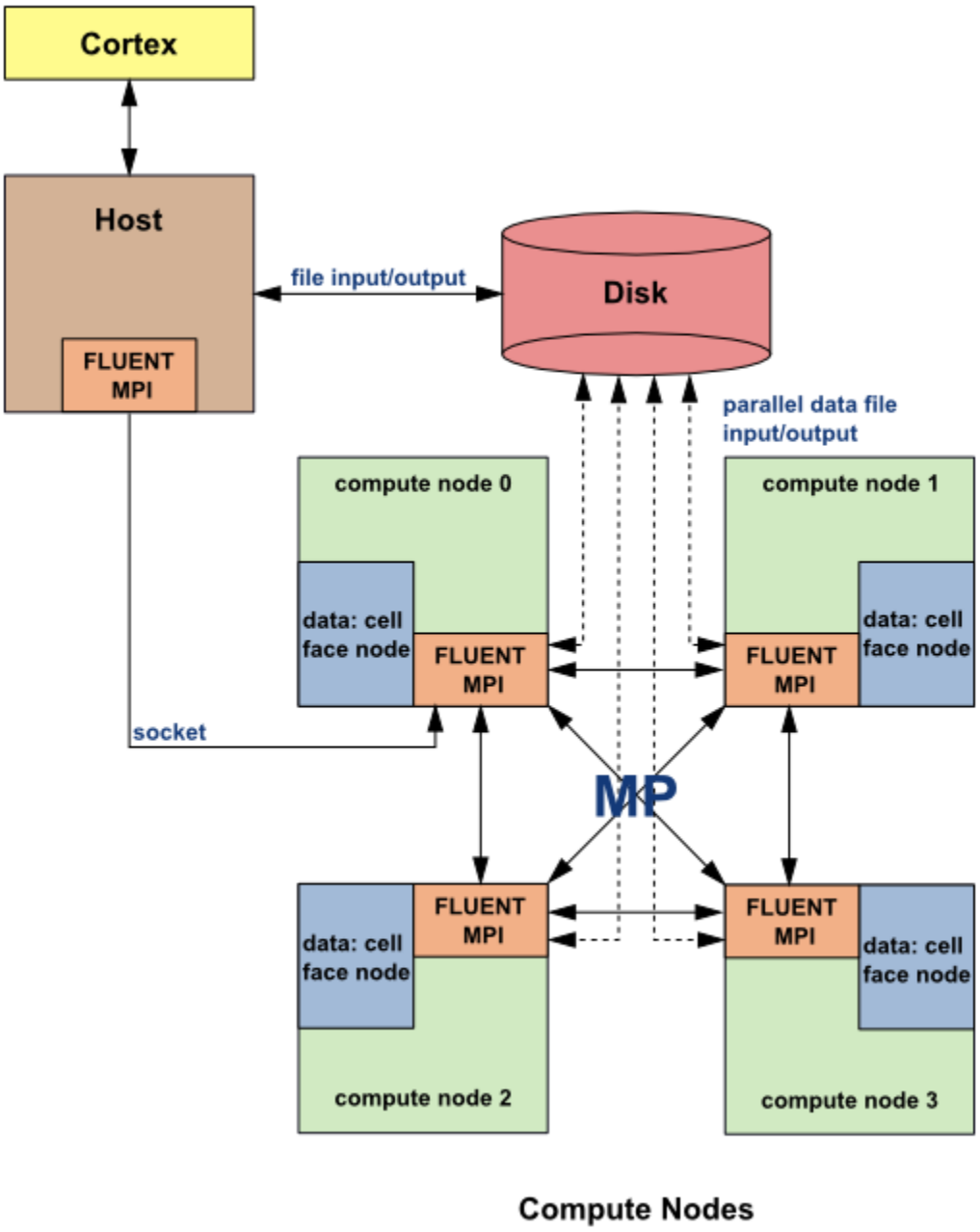
Figure 34.1: Serial ANSYS Fluent Architecture



ANSYS Fluent's parallel solver allows you to compute a solution by using multiple processes that may be executing on the same computer, or on different computers in a network (Figure 34.2: Parallel ANSYS Fluent Architecture (p. 2010)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 34.2: Parallel ANSYS Fluent Architecture



Parallel processing in ANSYS Fluent involves an interaction between ANSYS Fluent, a host process, and a set of compute-node processes. ANSYS Fluent interacts with the host process and the collection of compute nodes using a utility called `cortex` that manages ANSYS Fluent's user interface and basic graphical functions.

Parallel ANSYS Fluent splits up the mesh and data into multiple partitions, then assigns each mesh partition to a different compute process (or node). The number of partitions is equal to or less than the number of processors (or cores) available on your compute cluster. The compute-node processes can be executed on a massively-parallel computer, a multiple-CPU workstation, or a network cluster of computers.

Generally, as the number of compute nodes increases, turnaround time for solutions will decrease. This is referred to as solver "scalability." However, beyond a certain point, the ratio of network communication

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

to computation increases, leading to reduced parallel efficiency, so optimal system sizing is important for simulations.

ANSYS Fluent uses a host process that does not store any mesh or solution data. Instead, the host process only interprets commands from ANSYS Fluent's graphics-related interface, `cortex`.

The host distributes those commands to the other compute nodes via a socket interconnect to a single designated compute node called `compute-node-0`. This specialized compute node distributes the host commands to the other compute nodes. Each compute node simultaneously executes the same program on its own data set. Communication from the compute nodes to the host is possible only through `compute-node-0` and only when all compute nodes have synchronized with each other.

Each compute node is virtually connected to every other compute node, and relies on inter-process communication to perform such functions as sending and receiving arrays, synchronizing, and performing global operations (such as summations over all cells). Inter-process communication is managed by a message-passing library. For example, the message-passing library could be a vendor implementation of the Message Passing Interface (MPI) standard, as depicted in Figure 34.2: Parallel ANSYS Fluent Architecture (p. 2010).

All of the parallel ANSYS Fluent processes (as well as the serial process) are identified by a unique integer ID. The host collects messages from `compute-node-0` and performs operations (such as printing, displaying messages, and writing to a file) on all of the data, in the same way as the serial solver. You have the option of bypassing the host when inputting or outputting parallel data files, so that the data is passed directly between the compute nodes and the disk in a parallel fashion. This can reduce the time for data file I/O operations (for details, see Reading and Writing Parallel Data Files (p. 51)).

For additional information, see the following section:
34.1.1. Recommended Usage of Parallel ANSYS Fluent

34.1.1. Recommended Usage of Parallel ANSYS Fluent

The recommended procedure for using parallel ANSYS Fluent is as follows:

1. Start up the parallel solver. For details, see Starting Parallel ANSYS Fluent on a Windows System (p. 2021) and Starting Parallel ANSYS Fluent on a Linux System (p. 2026).

2. Read your case file and have ANSYS Fluent partition the mesh automatically upon loading it.

3. Review the partitions and perform partitioning again, if necessary. See Checking the Partitions (p. 2052) for details on checking your partitions. Note that there are other approaches for partitioning, including manual partitioning in either the serial or the parallel solver. For details, see Mesh Partitioning and Load Balancing (p. 2030).

4. Calculate a solution. See Checking and Improving Parallel Performance (p. 2060) for information on checking and improving the parallel performance.

Note

Due to limitations imposed by several MPI implementations, ANSYS Fluent performance on heterogeneous clusters involving either operating system or processor family differences may not be optimal, and in certain cases cause failures. You are urged to use caution in such parallel operating environments.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

34.2. Starting Parallel ANSYS Fluent Using Fluent Launcher

Whether you start ANSYS Fluent either from the Linux or Windows command line with no arguments, from the Windows Programs menu, or from the Windows desktop, Fluent Launcher will appear (for details, see Starting ANSYS Fluent Using Fluent Launcher in the Getting Started Guide), where you can specify the dimensionality of the problem (2D or 3D), as well as other options (for example, whether you want a single-precision or double-precision calculation).

Parallel calculation options can be set up by selecting **Parallel** under **Processing Options** in Fluent Launcher. Once you select the **Parallel** option, you can also specify the number of processes using the **Processes** field under **Solver**.

If your machines are equipped with appropriate General Purpose Graphical Processing Units (GPGPUs) you can indicate that these should be used for AMG solver acceleration by setting the **GPGPUs per Machine** option. Note that the number of solver processes per machine must be the same for all machines and that the number of processes per machine must be evenly divisible into the value you specify for **GPGPUs per Machine**. That is, for *nprocs* solver processes running on *M* machines using *ngpgpus* GPGPUS per machine:

$$nprocs \bmod M = 0$$
$$\left(\frac{nprocs}{M}\right) \bmod ngpgpus = 0$$

Table 34.1: Examples for GPGPUs per Machine (p. 2012) presents several examples illustrating the relationship between number of machines, number of solver processes, and GPGPUs per machine.

Table 34.1: Examples for GPGPUs per Machine

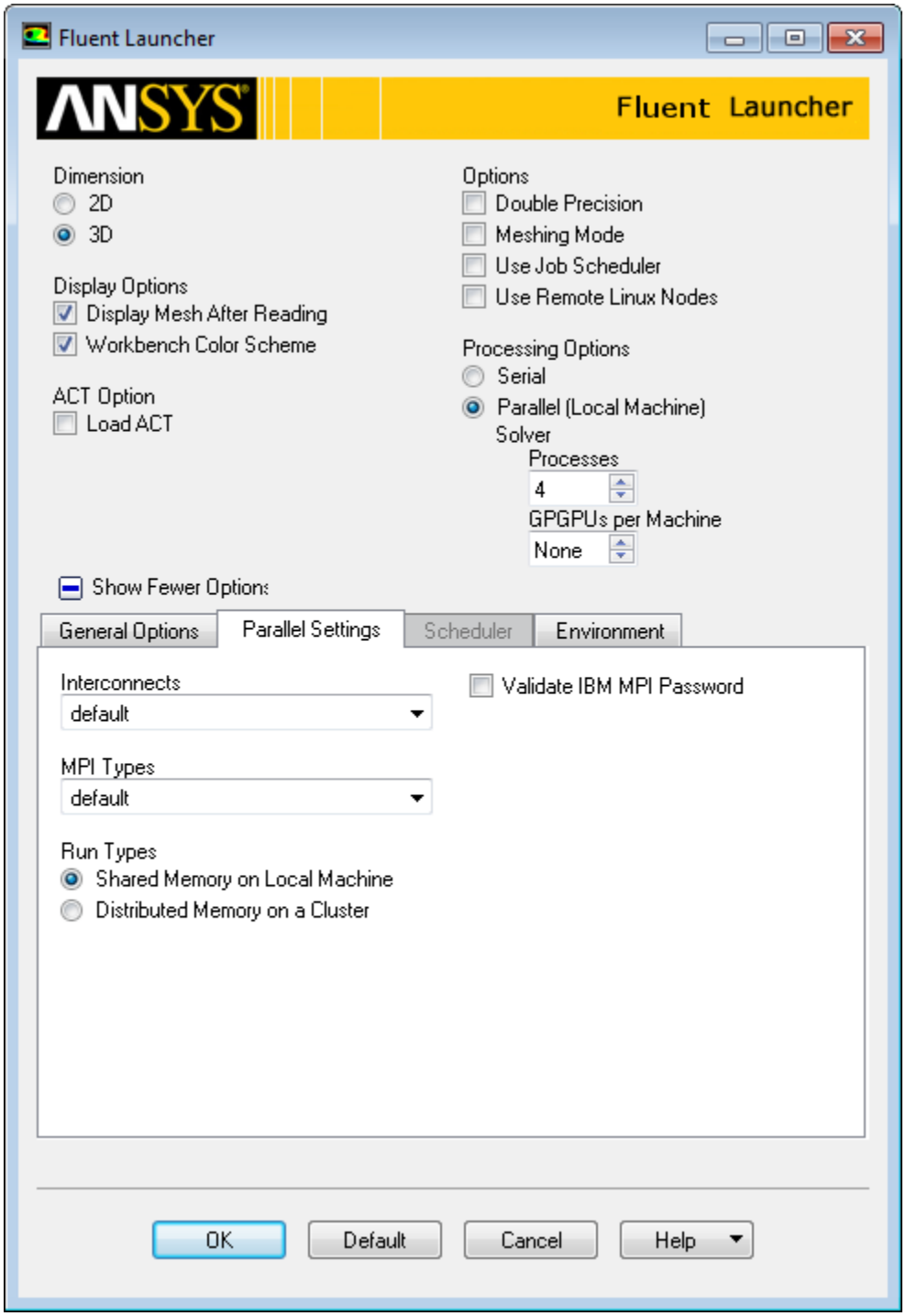
	Example 1	Example 2	Example 3
Number of Machines (`M`)	1	4	4
Number of Solver Processes (`nprocs`)	4	12	22
Valid values for GPGPUs per Machine (`ngpgpus`)	1, 2, 4	1, 3	`ngpgpus` will be ignored and GPGPU acceleration will be disabled (`M` does not evenly divide `nprocs`)

See Using General Purpose Graphics Processing Units (GPGPUs) With the Algebraic Multigrid (AMG) Solver (p. 2056) for more information about using GPGPU acceleration.

Activating the **Parallel** option enables the **Parallel Settings** tab (visible when you select the **Show More Options** button). The **Parallel Settings** tab allows you to specify settings for running ANSYS Fluent in parallel.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 34.3: The Parallel Settings Tab of Fluent Launcher



- Specify the interconnect in the **Interconnects** drop-down list. The default setting is recommended. For a symmetric multi-processor (SMP) system, the default setting uses shared memory for communication. On Windows, the best available interconnect is automatically used.

  (Linux only) If you prefer to select a specific interconnect, you can choose either **ethernet**, **myrinet**, or **infiniband**. For more information about these interconnects, see Table 34.5: Supported Interconnects for Linux Platforms (Per Platform) (p. 2028), Table 34.6: Available MPIs for Linux Platforms (p. 2028), and Table 34.7: Supported MPIs for Linux Architectures (Per Interconnect) (p. 2028).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Specify the type of message passing interface (MPI) you require for the parallel computations in the **MPI Types** field. The list of MPI types varies depending on the selected release and the selected architecture. There are several options, based on the operating system of the parallel cluster. For more information about the available MPI types, see Table 34.2: Supported Interconnects for the Windows Platform (p. 2023) - Table 34.3: Available MPIs for Windows Platforms (p. 2023).

  Important

  It is your responsibility to make sure the interconnects and the MPI types are compatible. If incompatible inputs are used, Fluent Launcher resorts to using the default values.

- (Linux Only) Specify either **RSH** (remote shell client) or **SSH** (secure shell client) under **Remote Spawn Command**. For more information about setting up your remote shell clients and secure shell clients, see Setting Up Your Remote Shell and Secure Shell Clients (p. 2029).

- Specify the type of parallel calculation under **Run Types**:

  – Select **Shared Memory on Local Machine** if the parallel calculations are performed by sharing memory allocations on your local machine.

  – Select **Distributed Memory on a Cluster** if the parallel calculations will be distributed among several machines.

    You can select **Machine Names** and enter the machine names directly into the text field as a list. Machine names can be separated either by a comma or a space. This is not recommended for a long list of machine names.

    Alternatively, you can select **File Containing Machine Names** to specify a hosts file (a file that contains the machine names), or you can use the button to browse for a hosts file.

    To edit an existing hosts file, click the button.

    By default, Fluent allocates ranks to machines in contiguous blocks, where the block sizes are as equal as possible. You can control the per machine block allocation size using the `machine` *X* `:` *Y* convention in the hosts specification, where *Y* is the process block count for `machine` *X*. The process assignment will cycle through the machine list until all processes are allocated in specified blocks. A fully round-robin assignment of processes can be achieved by setting the machine block allocation sizes to 1 (for example, `machine1:1`, `machine2:1`, and so on).

- Specify if you would like to validate the password for the IBM MPI or not.

  Select the **Validate IBM MPI Password** option if you would like to save the required password to use the IBM MPI type.

- For certain platforms, select **Use Job Scheduler** under **Options** if the parallel calculations are to be performed using a designated Job Scheduler (for details, see Setting Parallel Scheduler Options in Fluent Launcher (p. 2015)). This also enables the **Scheduler** tab of Fluent Launcher.

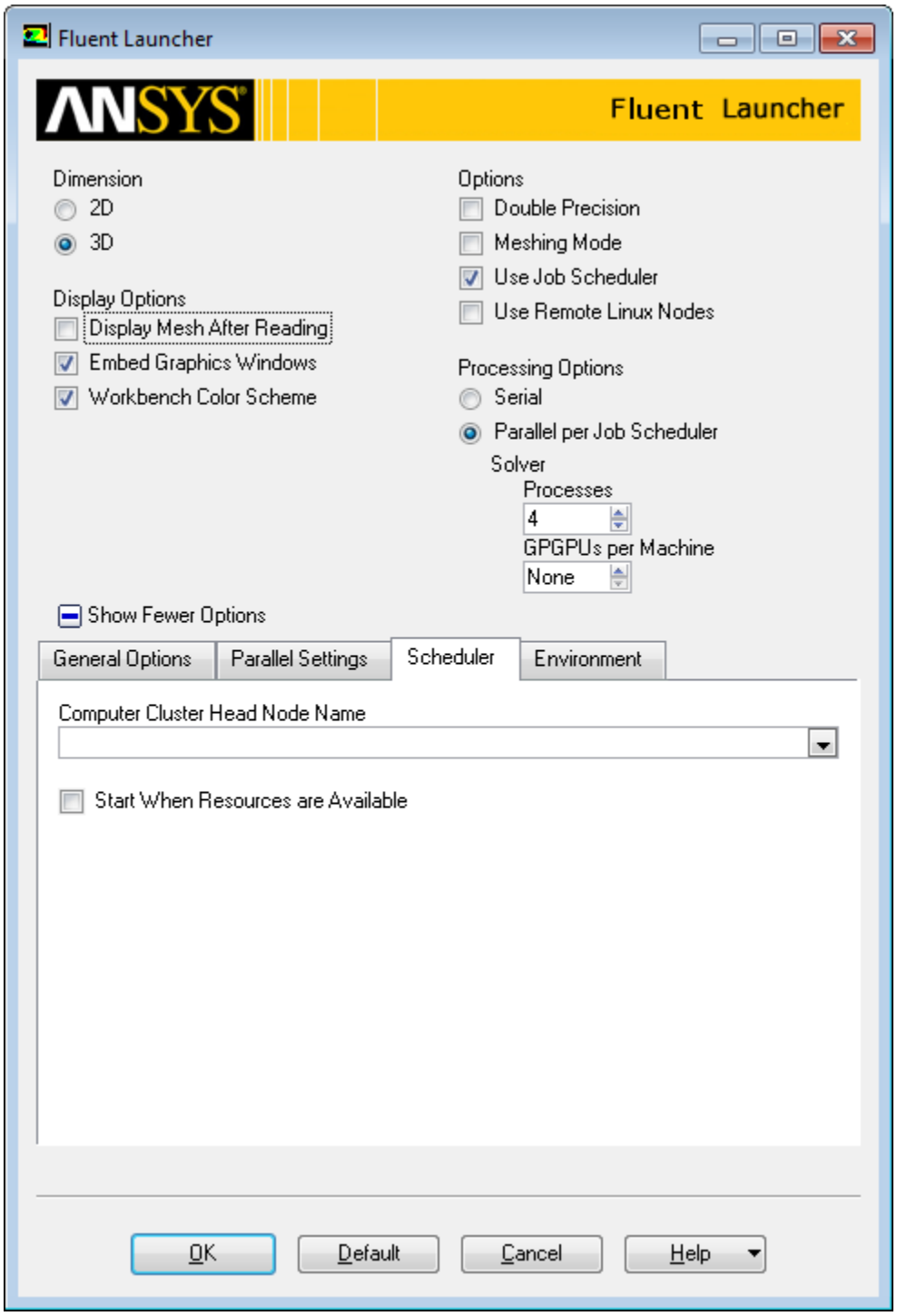
For additional information, see the following sections:
34.2.1. Setting Parallel Scheduler Options in Fluent Launcher
34.2.2. Setting Additional Options When Running on Remote Linux Machines

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

34.2.1. Setting Parallel Scheduler Options in Fluent Launcher

Activating the **Use Job Scheduler** option under **Options** in Fluent Launcher enables the **Scheduler** tab (visible when you select **Show More Options**). The **Scheduler** tab allows you to specify settings for running ANSYS Fluent with various job schedulers (for example, the Microsoft Job Scheduler for Windows, or LSF, SGE, and PBS Pro on Linux).

Figure 34.4: The Scheduler Tab of Fluent Launcher (Windows 64 Version)



For Windows 64-bit, with MSMPI or when the **Use Remote Linux Nodes** option is selected, you can specify that you want to use the Job Scheduler by selecting the **Use Job Scheduler** check box under

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Options in Fluent Launcher. Once selected, you can then enter a machine name in the **Compute Cluster Head Node Name** text field in the **Scheduler** tab. If you are running ANSYS Fluent on the head node, then you can keep the field empty. This option translates into the proper parallel command line syntax for using the Microsoft Job Scheduler (for details, see Starting Parallel ANSYS Fluent with the Microsoft Job Scheduler (p. 2023)).

If you want ANSYS Fluent to start after the necessary resources have been allocated by the Scheduler, then select the **Start When Resources are Available** check box.

For Linux, select the **Use Job Scheduler** check box under **Options** to use one of three available job schedulers in the **Scheduler** tab.

- Select **Use LSF** to use the LSF load management system with or without checkpointing. If you select **Use Checkpointing**, then you can specify a checkpointing directory in the **Checkpointing Directory** field. By default, the current working directory is used. In addition, you can specify a numerical value for the frequency of automatic checkpointing in the **Automatic Checkpoint with Setting of Period** field.

  For more information, see Setting Job Scheduler Options When Running on Remote Linux Machines (p. 2020) or Part I: Running ANSYS Fluent Under LSF.

- Select **Use SGE** to use the SGE load management system. You can choose to set values for the **SGE qmaster**, as well as the **SGE queue**, or the **SGE pe**. Alternatively, you can select **Use SGE settings** check box and specify the location and name of the SGE configuration file.

  For more information, see Setting Job Scheduler Options When Running on Remote Linux Machines (p. 2020) or Part III: Running ANSYS Fluent Under SGE.

- Select **Use PBS Pro** to use the PBS Pro load management system. You can choose to set the value for **PBS Submission Host** to specify the PBS Pro submission host name for submitting the job, if the machine you are using to run the launcher cannot submit jobs to PBS Pro.

  For more information, see Setting Job Scheduler Options When Running on Remote Linux Machines (p. 2020) or Part II: Running ANSYS Fluent Under PBS Professional.

For Windows, you also have the ability to run in batch mode (using the **Run in Batch Mode** check box) when you provide a journal file (designated in the **General Options** tab) that exits ANSYS Fluent at the end of the run.

For machines running the Windows HPC 2008 Server Scheduler, you also have the following options to choose from:

- **Job Template** allows you to create a custom submission policy to define the job parameters for an application. The cluster administrator can use job templates to manage job submission and optimize cluster usage.

- **Node Group** allows you to specify a collection of nodes. Cluster administrators can create groups and assign nodes to one or more groups.

- **Processor Unit** allows you to choose the following:

  – **Core** refers to a single computing unit in a machine. For example, a quad-core processor has 4 cores.

  – **Socket** refers to a set of tightly integrated cores as on a single chip. Machines often have 2 or more sockets, each socket with multiple cores. A dual CPU, hexcore processor, for example, having a total of 12 cores.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **Node** refers to a named host, that is, a single machine used as part of a cluster. Typical clusters range from a few to tens, hundreds, or sometimes thousands of machines.

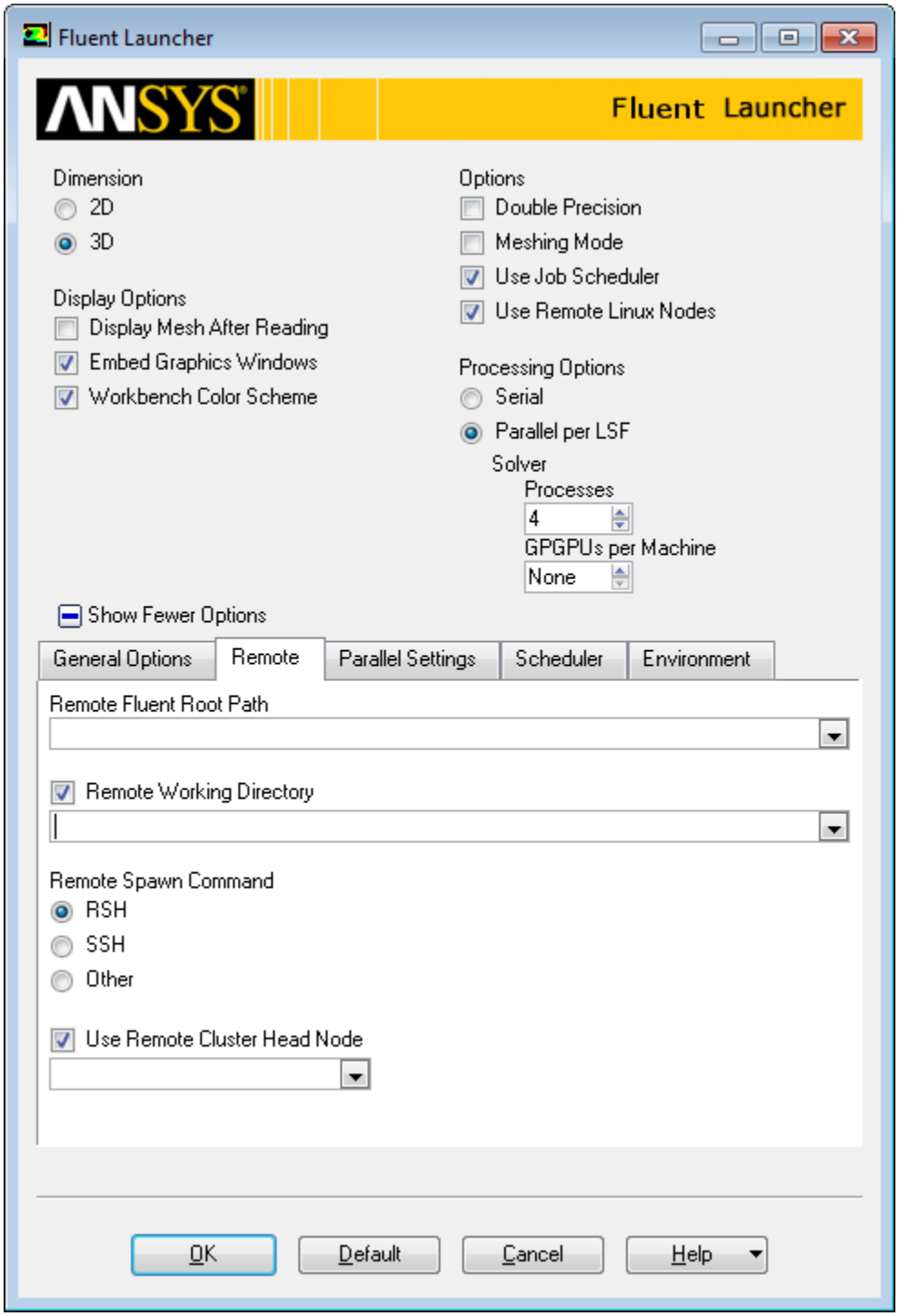
For more information about running Fluent jobs using the Windows HPC 2008 Server Scheduler, see the Frequently Asked Questions section of the Customer Portal.

34.2.2. Setting Additional Options When Running on Remote Linux Machines

The **Remote** tab allows you to specify settings for running ANSYS Fluent parallel simulations on Linux clusters, via the Windows interface.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 34.5: The Remote Tab of Fluent Launcher



You can run simulations on Linux machines, either in serial or on parallel Linux clusters, via the Windows interface. To access remote 64-bit Linux clusters for your parallel calculation, select the **Parallel (Local Machine)** option under **Processing Options** (for details, see Setting Parallel Options in Fluent Launcher in the Getting Started Guide), then enable **Use Remote Linux Nodes**, which appears under **Options**. The **Remote** tab in Fluent Launcher will become available, where you can specify the remote ANSYS Fluent Linux installation root path in the **Remote Fluent Root Path** field. The **Remote Working Directory** field allows you to specify a working directory for the remote Linux nodes, other than the default `temp` directory.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select one of the following **Remote Spawn Commands** to connect to the remote node:

- **RSH** will use RSH to spawn nodes from the local Windows machine to the Linux head node as well as from the Linux head node to the compute nodes. If you want the Linux cluster to use SSH, then you must set the `FLUENT_NO_REMOTE_RSH` to `1`. You also must set up password-less access.
- **SSH** (the default) will use SSH to spawn nodes from the local Windows machine to the Linux head node as well as from the Linux head node to the compute nodes. To use SSH with ANSYS Fluent, you must set up password-less SSH access. If you want the Linux cluster to use RSH, then you must set the `FLUENT_NO_REMOTE_SSH` to `1`. For more information about setting up SSH without a password, see www.debian-administration.org/articles/152.
- **Other** allows you to provide other compatible remote shell commands.

Enable the **Use Remote Cluster Head Node** field and specify the remote node to which ANSYS Fluent will connect for spawning (for example, via `rsh` or `ssh`). If this is not provided, then ANSYS Fluent will try to use the first machine in the machine file. If SGE is chosen as the job scheduler, then the **SGE qmaster** will serve the same purpose. If **PBS Pro** is chosen as the job scheduler, then the host specified here should be the PBS Pro submission host.

In addition to using the settings in the **Remote** tab in Fluent Launcher, the following command line options are also available when starting ANSYS Fluent from the command line:

`-nodepath=path`
is the path on the remote machine where ANSYS Fluent is installed.

`-node0=machine name`
is the machine from which to launch other nodes.

`-nodehomedir=directory`
is the directory that becomes the current working directory for all the nodes. Additionally, this will be used as a scratch area for temporary files that are created on the nodes.

`-rsh=remote shell command`
is the command that will be used to launch executables remotely. This option defaults to `ssh.exe` but can point to any equivalent program. The form of this command should be that it should not wait for additional inputs such as passwords. For example, if you install SSH, and try to launch in mixed mode using `ssh`, the launch may fail unless you have set up a login for SSH without a password. For more information about setting up SSH without a password, see www.debian-administration.org/articles/152.

As there are known issues with launching ANSYS Fluent in mixed Windows/Linux mode from cygwin, it is recommended that you use the command prompt (`cmd.exe`).

While ANSYS Fluent case and data files are read from and written to the Windows machine while running mixed Windows/Linux simulations, parallel data files (`.pdat` files) are written by the nodes and will not be available on the Windows machine. Therefore, the path specified for writing `.pdat` files should be a Linux path (you can set node's home directory if you want to provide relative paths). If you use the graphical user interface to read and/or write `.pdat` files, the directory component of the path is ignored and only the file name is used to read and/or write the `.pdat` file relative to the node's current working directory.

When working with mixed Linux and Windows runs that employ user-defined functions (UDFs), you should keep in mind that the file that you have opened for reading/writing on the host machine will not be available on remote nodes and vice-versa. You may therefore have to transfer data present on

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the nodes to the host and write it from host, (or distribute the data from the host to the nodes after reading the data from the host).

34.2.2.1. Setting Job Scheduler Options When Running on Remote Linux Machines

By selecting the **Use Remote Linux Nodes** option and the **Use Job Scheduler** option in Fluent Launcher, you can set job scheduler options for the remote Linux machines you are accessing for your CFD analysis.

When these options are enabled in Fluent Launcher, you can use the **Scheduler** tab to set parameters for either **LSF**, **SGE**, or **PBS Pro** job schedulers. You can learn more about each of the schedulers by referring to the Load Management Documentation.

The following list describes the various controls that are available in the **Scheduler** tab:

Use LSF
allows you to use the LSF job scheduler.

LSF queue
allows you to specify a job queue and enter the queue name in the field.

Use Checkpointing
allows you to use checkpointing with LSF. By default, the checkpointing directory will be the current working directory; however, you have the option of enabling **Checkpointing Directory**.

Checkpointing Directory
allows you to specify a checkpointing directory that is different from the current working directory.

Automatic Checkpoint with Setting of Period
allows you to specify that the checkpointing is done automatically at a set time interval. Enter the period (in minutes) in the field, otherwise checkpointing will not occur unless you call the `bchkpnt` command.

Use SGE
allows you to use the SGE job scheduler.

SGE qmaster
is the machine in the SGE job submission host list. SGE will allow the **SGE qmaster** node to summon jobs. By default, **localhost** is specified for **SGE qmaster**. Note that the button allows you to check the job status.

SGE queue
is the queue where you want to submit your ANSYS Fluent jobs. Note that you can use the button to contact the **SGE qmaster** for a list of queues. Leave this field blank if you want to use the default queue.

SGE pe
is the parallel environment where you want to submit your ANSYS Fluent jobs. The parallel environment must be defined by an administrator. For more information about creating a parallel environment, refer to the SGE documentation. Leave this field blank if you want to use the default parallel environment.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use PBS Pro
allows you to use the PBS Pro job scheduler.

Important

While running on remote Linux machines using any one of the Job Scheduler options, if the submitted job is in the job queue because of unavailable requested resources, then the ANSYS Fluent graphical user interface will remain open until resources are available and the job starts running.

34.3. Starting Parallel ANSYS Fluent on a Windows System

You can run ANSYS Fluent on a Windows system using either the graphical user interface (for details, see Starting Parallel ANSYS Fluent Using Fluent Launcher (p. 2012)) or command line options (for details, see Starting Parallel ANSYS Fluent on a Windows System Using Command Line Options (p. 2021)).

Important

See the separate installation instructions for more information about installing parallel ANSYS Fluent for Windows. The startup instructions below assume that you have properly set up the necessary software, based on the appropriate installation instructions.

Additional information about installation issues can also be found in the Frequently Asked Questions section of the Customer Portal.

For additional information, see the following section:
34.3.1. Starting Parallel ANSYS Fluent on a Windows System Using Command Line Options

34.3.1. Starting Parallel ANSYS Fluent on a Windows System Using Command Line Options

To start the parallel version of ANSYS Fluent using command line options, you can use the following syntax in a Command Prompt window:

`fluent` *version* `-t` *nprocs* `[-gpgpu=` *ngpgpus* `]` `[-p` *interconnect* `]` `[-mpi=` *mpi_type* `]` `[-cnf=` *hosts* `]`

where

- *version* must be replaced by the version of ANSYS Fluent you want to run (`2d`, `3d`, `2ddp`, or `3ddp`).
- `-t` *nprocs* specifies the number of processes to use. When the `-cnf` option is present, the *hosts* argument is used to determine which machines to use for the parallel job. For example, if there are 8 machines listed in the hosts file and you want to run a job with 4 processes, set *nprocs* to 4 (that is, `-t4`) and ANSYS Fluent will use the first 4 machines listed in the hosts file. Note that this does not apply to the Compute Cluster Server (CCS). If the `-gpgpu` option is used, *nprocs* must be chosen such that the number of solver processes per machine is equal on all machines.
- `-p` *interconnect* (optional) specifies the type of interconnect. The `ethernet` interconnect is used by default if the option is not explicitly specified. See Table 34.2: Supported Interconnects for the Windows Plat-

form (p. 2023), Table 34.3: Available MPIs for Windows Platforms (p. 2023), and Table 34.4: Supported MPIs for Windows Architectures (Per Interconnect) (p. 2023) for more information.

- `-mpi=` *mpi_type* (optional) specifies the MPI implementation. If the option is not specified, the default MPI for the given interconnect (IBM MPI) will be used (the use of the default MPI is recommended). The available MPIs for Windows are shown in Table 34.3: Available MPIs for Windows Platforms (p. 2023).

- `-cnf=` *hosts* (optional) specifies the hosts file, which contains a list of the machines on which you want to run the parallel job; if this option is not used, then the session will run on the local machine. If the hosts file is not located in the folder where you are typing the startup command, you must supply the full pathname to the file.

  You can use a plain text editor such as Notepad to create the hosts file. The only restriction on the filename is that there should be no spaces in it. For example, `hosts.txt` is an acceptable hosts file name, but `my hosts.txt` is not.

  Your hosts file (for example, `hosts.txt`) might contain the following entries:

  ```
  machine1
  machine2
  ```

  Important

  The last entry must be followed by a blank line.

  If a machine in the network is a multiprocessor, you can list it more than once. For example, if `machine1` has 2 CPUs, then, to take advantage of both CPUs (and similarly for multicore machines), the `hosts.txt` file should list `machine1` twice:

  ```
  machine1
  machine1
  machine2
  ```

  By default, Fluent allocates ranks to machines in contiguous blocks, where the block sizes are as equal as possible. You can control the per machine block allocation size using the `machine` *X* `:` *Y* convention in the hosts specification, where *Y* is the process block count for `machine` *X*. The process assignment will cycle through the machine list until all processes are allocated in specified blocks. A fully round-robin assignment of processes can be achieved by setting the machine block allocation sizes to 1 (for example, `machine1:1`, `machine2:1`, and so on).

- `-gpgpu=` *ngpgpus* specifies the number of GPGPUs per machine to use for AMG execution. Note that when this option is used, the number of solver process per machine must be equal on all machines and *ngpgpus* must be chosen such that the number of solver processes per machine is an integer multiple of *ngpgpus*. That is, for *nprocs* solver processes running on *M* machines using *ngpgpus* GPGPUS per machine:

  $$nprocs \bmod M = 0$$
  $$\left(\frac{nprocs}{M}\right) \bmod ngpgpus = 0$$

  See Using General Purpose Graphics Processing Units (GPGPUs) With the Algebraic Multigrid (AMG) Solver (p. 2056) for more information about using GPGPU acceleration.

For example, the full command line to start a 3D parallel job on the first 4 machines listed in a hosts file called `hosts.txt` is as follows:

```
fluent 3d -t4 -cnf=hosts.txt
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

As another example, the full command line to start a 3D symmetrical multiprocessing (SMP) parallel job on 4 machines is as follows:

```
fluent 3d -t4
```

In either case, the default communication library (IBM MPI), and the default interconnect (automatically selected by the MPI used, or `ethernet`) will be used since these options are not specified.

The first time that you try to run ANSYS Fluent in parallel, you will be prompted for information about the current Windows account. For more information about the IBM MPI setup, see the ANSYS Customer Portal (www.ansys.com/customerportal).

The supported interconnects for dedicated parallel `win64` Windows machines, the associated MPIs for them, and the corresponding syntax are listed in Table 34.2: Supported Interconnects for the Windows Platform (p. 2023) - Table 34.4: Supported MPIs for Windows Architectures (Per Interconnect) (p. 2023).

Table 34.2: Supported Interconnects for the Windows Platform

Platform	Processor	Architecture	Interconnects
Windows	64-bit	`win64`	`ethernet` (default), `infiniband`, `myrinet`

Table 34.3: Available MPIs for Windows Platforms

MPI	Syntax (flag)	Communication Library	Notes
`ibmmpi`	`-mpi=ibmmpi`	IBM MPI	(1), (2), (3)
`ms`	`-mpi=ms`	Microsoft MPI	(1)*, (2)
`intel`	`-mpi=intel`	Intel MPI	(1), (2), (4)

(1) Used with Shared Memory Machine (SHM) where the memory is shared between the processors on a single machine.

* Ensure that Microsoft MPI is installed on the machine where the shared memory job will be running.

(2) Used with Distributed Memory Machine (DMM) where each processor has it's own memory associated with it.

(3) IBM Platform MPI Community Edition is installed with Fluent, which is limited to 4096 processes without an additional license. Refer to IBM Platform MPI with High (>4096) Process Count for details.

(4) By default, Intel MPI does not support heterogeneous environments (that is, a mix of operating system versions or hardware) for Windows. As a workaround, you can use the following environment setting:

```
I_MPI_PLATFORM zero
```

Table 34.4: Supported MPIs for Windows Architectures (Per Interconnect)

Architecture	Ethernet	Myrinet	Infiniband
`win64`	`ibmmpi` (default), `intel`, `ms`	`ms`	`ms`

34.3.1.1. Starting Parallel ANSYS Fluent with the Microsoft Job Scheduler

The Microsoft Job Scheduler allows you to manage multiple jobs and tasks, allocate machine resources, send tasks to compute nodes, and monitor jobs, tasks, and compute nodes.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Windows Server operating systems include a compute cluster server (CCS) and a high performance computing server (HPC) that combines the Microsoft MPI type (`msmpi`) with the Microsoft Job Scheduler. ANSYS Fluent provides a means of using the Microsoft Job Scheduler using the following flag in the parallel command:

`-ccp` *head-node-name*

where `-ccp` indicates the use of the compute cluster server package, and *head-node-name* indicates the name of the head node of the machine cluster.

For example, if you want to use the Microsoft Job Scheduler to run a 3D model on 2 nodes, the corresponding command syntax would be:

`fluent 3d -t2 -ccp` *head-node-name*

Important

Both the IBM MPI type (`ibmmpi`) and the Intel MPI (`intel`) are not supported with the Microsoft Job Scheduler.

Note

When using Microsoft Job Scheduler, the best interconnect is automatically selected by MS-MPI and the default Ethernet option does not apply.

Though the usage described previously is recommended as an initial starting point for running ANSYS Fluent with the Microsoft Job Scheduler, there are further options provided to meet your specific needs. ANSYS Fluent allows you to do any of the following with the Microsoft Job Scheduler:

- Request resources from the Microsoft Job Scheduler first, before you launch ANSYS Fluent.

  This is done by first submitting a job that will run until canceled, as shown in the following example:

  `job new/scheduler:` *head-node-name* `/numprocessors:2 /rununtilcanceled:true`

  This example requests a 2-node resource on a cluster named *head-node-name*. You will see that a job is created with the job ID *job-id*:

  `job submit/scheduler:` *head-node-name* `/id:` *job-id*

  Then check if the resources have been allocated:

  `job view` *job-id* `/scheduler:` *head-node-name*

  If the resources are ready, you can start ANSYS Fluent using the job ID:

  `fluent 3d -t2 -ccp` *head-node-name* `-jobid=` *job-id*

  This job will be reusable until you decide to cancel it, at which point you must enter the following:

  `job cancel` *job-id* `/scheduler:` *head-node-name*

- Have ANSYS Fluent submit a CCS job, but delay the launching of ANSYS Fluent until the actual resources are allocated.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This is done by specifying the job ID as `-1`, as shown in the following example:

`fluent 3d -t2 -ccp` *head-node-name* `-jobid=-1`

If you want to stop the job application, click the **Cancel** button. ANSYS Fluent will prompt you for confirmation, and then clean up the pending job and exit.

- Run your job using XML template files.

  This is done by first creating an XML template file, such as shown in the following example:

```
<?xml version="1.0" encoding="utf-8"?>
<Job
xmlns:m:xsi="http://www.w3.org/2001/XMLSchema-instance"
xmlns:m:xsd="http://www.w3.org/2001/XMLSchema"
SoftwareLicense=""
MaximumNumberOfProcessors="4"
MinimumNumberOfProcessors="4"
Runtime="Infinite"
IsExclusive="true"
Priority="Normal"
Name="Name_of_job"
Project="Fluent runs"
RunUntilCanceled="false">
<Tasks xmlns:m="http://www.microsoft.com/ComputeCluster/">
 <Task
   MaximumNumberOfProcessors="4"
   MinimumNumberOfProcessors="4"
   Depend=""
   WorkDirectory="\\file-server\home\user"
   Stdout="fluent-case.%CCP_JOBID%.out"
   Stderr="fluent-case.%CCP_JOBID%.err"
   Name="My Task"
   CommandLine="\\head-node\fluent-sharename\win64\fluent.exe 3d -i bsi.jou -t4"
   IsExclusive="true"
   IsRerunnable="false"
   Runtime="Infinite">
 </Task>
</Tasks>
</Job>
```

  where `fluent-sharename` is the name of the shared directory pointing to where ANSYS Fluent is installed (for example, `C:\Program Files\ANSYS Inc\v181\fluent`).

  Important

  Note that you must create a journal file that exits ANSYS Fluent at the end of the run, and refer to it using the `-i` flag in your XML template file (`bs1.jou` in the previous example).

  After you have saved the file and given it a name (for example, `job1.xml`), you can submit the job as shown:

  `job submit /jobfile:job1.xml`

- Run the job in batch mode without displaying the ANSYS Fluent graphical user interface.

  The following is an example of such a batch mode job:

  `job submit /scheduler:` *head-node-name*

  `/numprocessors:2 /workdir:\\file-server\home\user\`

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
\\head-node\fluent-sharename\ntbin\win64\fluent.exe 3d -t2 -i bs1.jou
```

where `fluent-sharename` is the name of the shared directory pointing to where ANSYS Fluent is installed (for example, `C:\Program Files\ANSYS Inc\v181\fluent`).

Important

- Note that you must create a journal file that exits ANSYS Fluent at the end of the run, and refer to it using the `-i` flag in your batch mode job submission (`bs1.jou` in the previous example).
- You can start ANSYS Fluent jobs from any machine on which is installed either the full CCP or the CCP client tools, but note that all the machines must have the same version installed.

34.4. Starting Parallel ANSYS Fluent on a Linux System

You can run ANSYS Fluent on a Linux system using either the graphical user interface (for details, see Starting Parallel ANSYS Fluent Using Fluent Launcher (p. 2012)) or command line options (for details, see Starting Parallel ANSYS Fluent on a Linux System Using Command Line Options (p. 2026) and Setting Up Your Remote Shell and Secure Shell Clients (p. 2029)).

For additional information, see the following sections:

34.4.1. Starting Parallel ANSYS Fluent on a Linux System Using Command Line Options
34.4.2. Setting Up Your Remote Shell and Secure Shell Clients

34.4.1. Starting Parallel ANSYS Fluent on a Linux System Using Command Line Options

To start the parallel version of ANSYS Fluent using command line options, you can use the following syntax in a command prompt window:

`fluent` *version* `-t` *nprocs* `[-gpgpu=` *ngpgpus* `] [-p` *interconnect* `] [-mpi=` *mpi_type* `] [-cnf=` *hosts* `]`

where

- *version* must be replaced by the version of ANSYS Fluent you want to run (`2d`, `3d`, `2ddp`, or `3ddp`).
- `-t` *nprocs* specifies the number of processes to use. When the `-cnf` option is present, the *hosts* argument is used to determine which machines to use for the parallel job. For example, if there are 10 machines listed in the hosts file and you want to run a job with 5 processes, set *nprocs* to 5 (that is, `-t5`) and ANSYS Fluent will use the first 5 machines listed in the hosts file. Note that if the `--gpgpu` option is used, *nprocs* must be chosen such that the number of solver processes per machine is equal on all machines.
- `-p` *interconnect* (optional) specifies the type of interconnect. The `auto-select` interconnect is used by default so that the best available interconnect is used if the option is not explicitly specified. See Table 34.5: Supported Interconnects for Linux Platforms (Per Platform) (p. 2028), Table 34.6: Available MPIs for Linux Platforms (p. 2028), and Table 34.7: Supported MPIs for Linux Architectures (Per Interconnect) (p. 2028) for more information.
- `-mpi=` *mpi_type* (optional) specifies the type of MPI. If the option is not specified, the default MPI for the given interconnect will be used (the use of the default MPI is recommended). The available MPIs for Linux are shown in Table 34.6: Available MPIs for Linux Platforms (p. 2028).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `-cnf=` *hosts* (optional) specifies the hosts file, which contains a list of the machines on which you want to run the parallel job; if this option is not used, then the session will run on the local machine. If the hosts file is not located in the directory where you are typing the startup command, you must supply the full pathname to the file.

  You can use a plain text editor to create the hosts file. The only restriction on the filename is that there should be no spaces in it. For example, `hosts.txt` is an acceptable hosts file name, but `my hosts.txt` is not.

  Your hosts file (for example, `hosts.txt`) might contain the following entries:

  ```
  machine1
  machine2
  ```

  If a machine in the network is a multiprocessor, you can list it more than once. For example, if `machine1` has 2 CPUs, then, to take advantage of both CPUs, the `hosts.txt` file should list `machine1` twice:

  ```
  machine1
  machine1
  machine2
  ```

  As an alternative to a hosts file, you can simply type the names of the machines in a list separated by commas, as shown in the following example: `-cnf=machine1,machine1,machine2`.

  By default, Fluent allocates ranks to machines in contiguous blocks, where the block sizes are as equal as possible. You can control the per machine block allocation size using the `machine` *X* `:` *Y* convention in the hosts specification, where *Y* is the process block count for `machine` *X*. The process assignment will cycle through the machine list until all processes are allocated in specified blocks. A fully round-robin assignment of processes can be achieved by setting the machine block allocation sizes to 1 (for example, `machine1:1`, `machine2:1`, and so on).

- `-gpgpu=` *ngpgpus* specifies the number of GPGPUs per machine to use for AMG execution. Note that when this option is used, the number of solver process per machine must be equal on all machines and *ngpgpus* must be chosen such that the number of solver processes per machine is an integer multiple of *ngpgpus*. That is, for *nprocs* solver processes running on *M* machines using *ngpgpus* GPGPUS per machine:

  $$nprocs \bmod M = 0$$
  $$\left(\frac{nprocs}{M}\right) \bmod ngpgpus = 0$$

  See Using General Purpose Graphics Processing Units (GPGPUs) With the Algebraic Multigrid (AMG) Solver (p. 2056) for more information about using GPGPU acceleration.

For example, to use the Infiniband interconnect, and to start the 3D solver with 4 compute nodes on the machines defined in the text file called `fluent.hosts`, you can enter the following in the command prompt:

```
fluent 3d -t4 -pinfiniband -cnf=fluent.hosts
```

Note that if the optional `-cnf=` *hosts* is specified, a compute node will be spawned on each machine listed in the file *hosts*.

Also, ANSYS Fluent provides a fault-tolerance feature on Infiniband Linux clusters running OFED. To invoke this feature, use the command line flag `-pinfiniband.ofedft` (or `-pib.ofedft`), which enables transparent port fail-over and high-availability features using the IBM MPI. Note that while the simulations proceed more robustly with this option, there may be some degradation in performance.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

During startup, ANSYS Fluent will create very small text files in the `/tmp` area. If you would like to specify a different location for these files, set the following environment variable: `export FL_TMPDIR=` *directory*.

The supported interconnects for parallel Linux machines are listed below (Table 34.5: Supported Interconnects for Linux Platforms (Per Platform) (p. 2028), Table 34.6: Available MPIs for Linux Platforms (p. 2028), and Table 34.7: Supported MPIs for Linux Architectures (Per Interconnect) (p. 2028)), along with their associated communication libraries, the corresponding syntax,

Table 34.5: Supported Interconnects for Linux Platforms (Per Platform)

Platform	Processor	Architecture	Interconnects/Systems*
Linux	64-bit	`lnamd64`	`ethernet`, `infiniband`, `myrinet`

(*) Node processes on the same machine communicate by shared memory. ANSYS Fluent lets the MPI autoselect the best interconnect available on the system. Users can specify an interconnect to override that selection. Ethernet is the fallback choice.

Table 34.6: Available MPIs for Linux Platforms

MPI	Syntax (flag)	Communication Library	Notes
`ibmmpi`	`-mpi=ibmmpi`	IBM MPI	General purpose for SMPs and clusters (1)
`intel`	`-mpi=intel`	Intel MPI	General purpose for SMPs and clusters
`openmpi`	`-mpi=openmpi`	Open MPI	Open source MPI-2 implementation. For both SMPs and clusters.
`cray`	`-mpi=cray`	Cray MPI (MPT)	Only available on Cray systems.

(1) IBM Platform MPI Community Edition is installed with Fluent, which is limited to 4096 processes without an additional license. Refer to IBM Platform MPI with High (>4096) Process Count for details.

Table 34.7: Supported MPIs for Linux Architectures (Per Interconnect)

Architecture	Ethernet	Myrinet*	Infiniband	Proprietary Systems	Omni-Path****
`lnamd64`	`ibmmpi` (default), `intel`, and `openmpi`	`ibmmpi` (default), `openmpi`**	`ibmmpi` (default), `intel`, and `openmpi`	Cray***	`intel`

(*) Both MX and GM Myrinet protocols are supported. ANSYS Fluent will automatically detect which type is running on the system and will use that particular protocol. You only have to supply the `-pmyrinet` option. If the hardware supports it, the installation and usage of Myrinet MX is recommended (consult your Myrinet vendor for applicability).

(**) The OpenMPI version packaged with ANSYS Fluent does not support Myrinet. However, you can use a local version of OpenMPI with Myrinet support.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(***) Using the Cray platform, the Extreme Scalability Mode (ESM) option is used by default, employing the native Cray MPI (MPT) library for parallel communication. Optionally, you can use the Cluster Compatibility Mode (CCM) mode, in which the standard IBM MPI library is used for parallel communication. For simulations with high core counts (for example, above 1000 cores), the ESM mode is recommended for better performance. Note that when using MPT version 5.0 and higher (up to, but not including 7.0), you must set the following environment variable: `export FLUENT_USE_CRAY_MPT5=1`.

(****) Omni-Path requires OFED driver version 10.2 or higher.

(Linux Only) To enable a job scheduler supported native remote node access mechanism, you can use the `-scheduler_rsh` command line flag or set the environment variable SCHEDULER_RSH=1. This tight integration is only supported for the MPI/Job scheduler combinations listed below:

- IBM MPI + LSF
- Intel MPI + LSF
- Intel MPI + UGE
- Intel MPI + PBS Pro

34.4.2. Setting Up Your Remote Shell and Secure Shell Clients

For cluster computing on Linux systems, most parallel versions of ANSYS Fluent will require the user account set up such that you can connect to all nodes on the cluster (using either the remote shell (`rsh`) client or the secure shell (`ssh`) client) without having to enter a password each time for each machine.

Provided that the appropriate server daemons (either `rshd` or `sshd`) are running, this section briefly describes how you can configure your system in order to use ANSYS Fluent for parallel computing.

34.4.2.1. Configuring the `rsh` Client

The remote shell client (`rsh`), is widely deployed and used. It is generally easy to configure, and involves adding all the machine names, each on a single line, to the `.rhosts` file in your home directory.

If you refer to the machine you are currently logged on as the 'client', and if you refer to the remote machine to which you seek password-less login as the 'server', then on the server, you can add the name of your client machine to the `.rhosts` file. The name could be a local name or a fully qualified name with the domain suffix. Similarly, you can add other clients from which you require similar access to this server. These machines are then "trusted" and remote access is allowed without the further need for a password. This setup assumes you have the same user ID on all the machines. Otherwise, each line in the `.rhosts` file must contain the machine name as well as the user ID for the client that you want access to. Refer to your system documentation for further usage options.

Note that for security purposes, the `.rhosts` file must be readable only by the user.

34.4.2.2. Configuring the `ssh` Client

The secure shell client (`ssh`), is a more secure alternative than `rsh` and is also used widely. Depending on the specific protocol and the version deployed, configuration involves a few steps. SSH1 and SSH2 are two current protocols. `OpenSSH` is an open implementation of the SSH2 protocol and is backwards compatible with the SSH1 protocol. To add a client machine, with respect to user configuration, the following steps are involved:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Generate a public-private key pair using `ssh-keygen` (or using a graphical user interface client). For example:

   ```
   % ssh-keygen -t dsa
   ```

   where it creates a Digital Signature Authority (DSA) type key pair.

2. Place your public key on the remote host.

   - For SSH1, insert the contents of the client (`~/.ssh/identity.pub`) into the server (`~/.ssh/authorized_keys`).
   - For SSH2, insert the contents of the client (`~/.ssh/id_dsa.pub`) into the server (`~/.ssh/authorized_keys2`).

The client machine is now added to the access list and you are no longer required to type in a password each time. For additional information, consult your system administrator or refer to your system documentation.

34.5. Mesh Partitioning and Load Balancing

Information about mesh partitioning and load balancing is provided in the following sections:

34.5.1. Overview of Mesh Partitioning
34.5.2. Partitioning the Mesh Automatically
34.5.3. Partitioning the Mesh Manually and Balancing the Load
34.5.4. Using the Partitioning and Load Balancing Dialog Box
34.5.5. Mesh Partitioning Methods
34.5.6. Checking the Partitions
34.5.7. Load Distribution
34.5.8. Troubleshooting

34.5.1. Overview of Mesh Partitioning

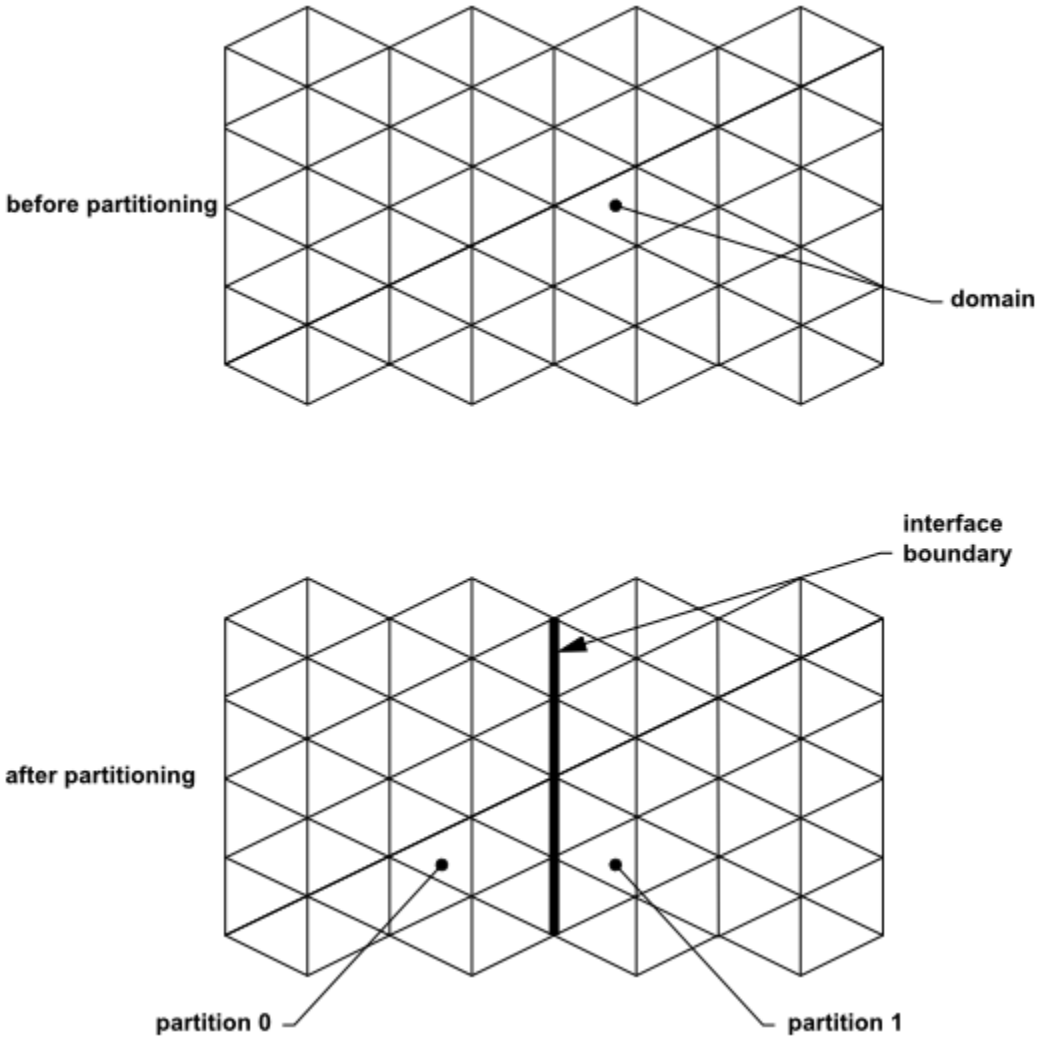
When you use the parallel solver in ANSYS Fluent, you must partition or subdivide the mesh into groups of cells that can be solved on separate processors (see Figure 34.6: Partitioning the Mesh (p. 2031)). You can either use the automatic partitioning algorithms when reading an unpartitioned mesh into the parallel solver (recommended approach, described in Partitioning the Mesh Automatically (p. 2031)), or perform the partitioning yourself in the serial solver or after reading a mesh into the parallel solver (as described in Partitioning the Mesh Manually and Balancing the Load (p. 2033)). In either case, the available partitioning methods are those described in Mesh Partitioning Methods (p. 2046). You can partition the mesh before or after you set up the problem (by defining models, boundary conditions, and so on).

Note that the relative distribution of cells among compute nodes will be maintained during mesh adaption, so manual repartitioning after adaption is not required. For details, see Load Distribution (p. 2055).

If you use the serial solver to set up the problem before partitioning, the machine on which you perform this task must have enough memory to read in the mesh. If your mesh is too large to be read into the serial solver, you can read the unpartitioned mesh directly into the parallel solver (using the memory available in all the defined hosts) and have it automatically partitioned. In this case you will set up the problem after an initial partition has been made. You will then be able to manually repartition the case if necessary. See Partitioning the Mesh Automatically (p. 2031) and Partitioning the Mesh Manually and Balancing the Load (p. 2033) for additional details and limitations, and Checking the Partitions (p. 2052) for details about checking the partitions.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 34.6: Partitioning the Mesh



34.5.2. Partitioning the Mesh Automatically

For automatic mesh partitioning, you can select the partition method and other options for creating the mesh partitions before reading a case file into the parallel version of the solver. For some of the methods, you can perform pretesting to ensure that the best possible partition is performed. See Mesh Partitioning Methods (p. 2046) for information about the partitioning methods available in ANSYS Fluent.

Note

Architecturally aware partitioning (see Partitioning (p. 2033)) is performed automatically when the case file is read. If the maximum inter-machine communication is reduced by more than 5%, the new partition mapping will be applied, and a message is displayed in the console, for example:

```
inter-node communication reduction by architecture-aware remapping:
47%
```

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

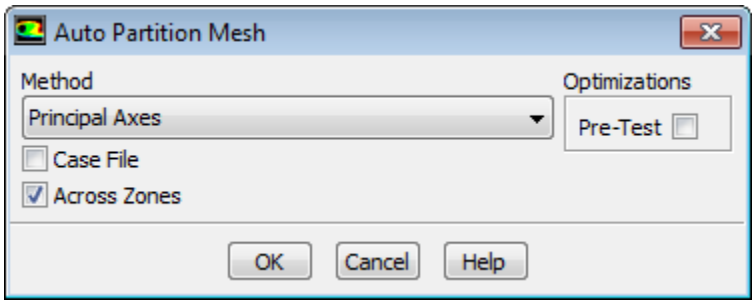
While the message indicates actual point-to-point network traffic reduction, solver computational performance improvement may be somewhat less, and depends on the case and the system network configuration.

The procedure for partitioning automatically in the parallel solver is as follows:

1. (optional) Set the partitioning parameters in the **Auto Partition Mesh** dialog box (Figure 34.7: The Auto Partition Mesh Dialog Box (p. 2032)).

   Parallel → General → Auto Partition...

   Figure 34.7: The Auto Partition Mesh Dialog Box

   If you are reading in a mesh file or a case file for which no partition information is available, and you keep the **Case File** option turned on, ANSYS Fluent will partition the mesh using the method displayed in the **Method** drop-down list.

   If you want to specify the partitioning method and associated options yourself, the procedure is as follows:

   a. Turn off the **Case File** option. The other options in the dialog box will become available.

   b. Select the partition method in the **Method** drop-down list. The choices are the techniques described in Partition Methods (p. 2046).

   c. You can choose to independently apply partitioning to each cell zone, or you can allow partitions to cross zone boundaries using the **Across Zones** check button. It is recommended that you *not* partition cells zones independently (by turning off the **Across Zones** check button) unless cells in different zones will require significantly different amounts of computation during the solution phase (for example, if the domain contains both solid and fluid zones).

   d. If you have chosen the **Principal Axes** or **Cartesian Axes** method, you can improve the partitioning by enabling the automatic testing of the different bisection directions before the actual partitioning occurs. To use pretesting, turn on the **Pre-Test** option. Pretesting is described in Pretesting (p. 2051).

   e. Click **OK**.

   If you have a case file where you have already partitioned the mesh, and the number of partitions divides evenly into the number of compute nodes, you can keep the default selection of **Case File** in the **Auto Partition Mesh** dialog box. This instructs ANSYS Fluent to use the partitions in the case file.

2. Read the case file.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File → Read → Case...

34.5.2.1. Reporting During Auto Partitioning

As the mesh is automatically partitioned, some information about the partitioning process will be displayed in the console. If you want additional information, you can display a report from the **Partitioning and Load Balancing** dialog box after the partitioning is completed.

Parallel → General → Partition/Load Balance...

When you click the **Print Active Partitions** or **Print Stored Partitions** button in the **Partitioning and Load Balancing** dialog box, ANSYS Fluent will display the partition ID, number of cells, faces, and interfaces, and the ratio of interfaces to faces for each active or stored partition in the console. In addition, it will display the minimum and maximum cell, face, interface, and face-ratio variations. For details, see Interpreting Partition Statistics (p. 2052). You can examine the partitions graphically by following the directions in Checking the Partitions (p. 2052).

34.5.3. Partitioning the Mesh Manually and Balancing the Load

Automatic partitioning in the parallel solver (described in Partitioning the Mesh Automatically (p. 2031)) is the recommended approach to mesh partitioning, but it is also possible to partition the mesh manually in either the serial solver or the parallel solver. After automatic or manual partitioning, you will be able to inspect the partitions created (for details, see Checking the Partitions (p. 2052)) and optionally repartition the mesh, if necessary. Again, you can do so within the serial or the parallel solver, using the **Partitioning and Load Balancing** dialog box. A partitioned mesh may also be used in the serial solver without any loss in performance.

34.5.3.1. Guidelines for Partitioning the Mesh

The following steps are recommended for partitioning a mesh manually:

1. Partition the mesh using the default method (**Metis**). Metis will generally produce the best quality partitions for most problems and no further user intervention should be necessary.

2. Examine the partition statistics, which are described in Interpreting Partition Statistics (p. 2052). Your aim is to achieve small magnitudes for Minimum, Maximum, and Total `Partition boundary face count ratio` while maintaining a balanced load (`Mean cell count variation`). If the statistics are not acceptable, try one of the other partition methods.

Instructions for manual partitioning are provided below.

34.5.4. Using the Partitioning and Load Balancing Dialog Box

34.5.4.1. Partitioning

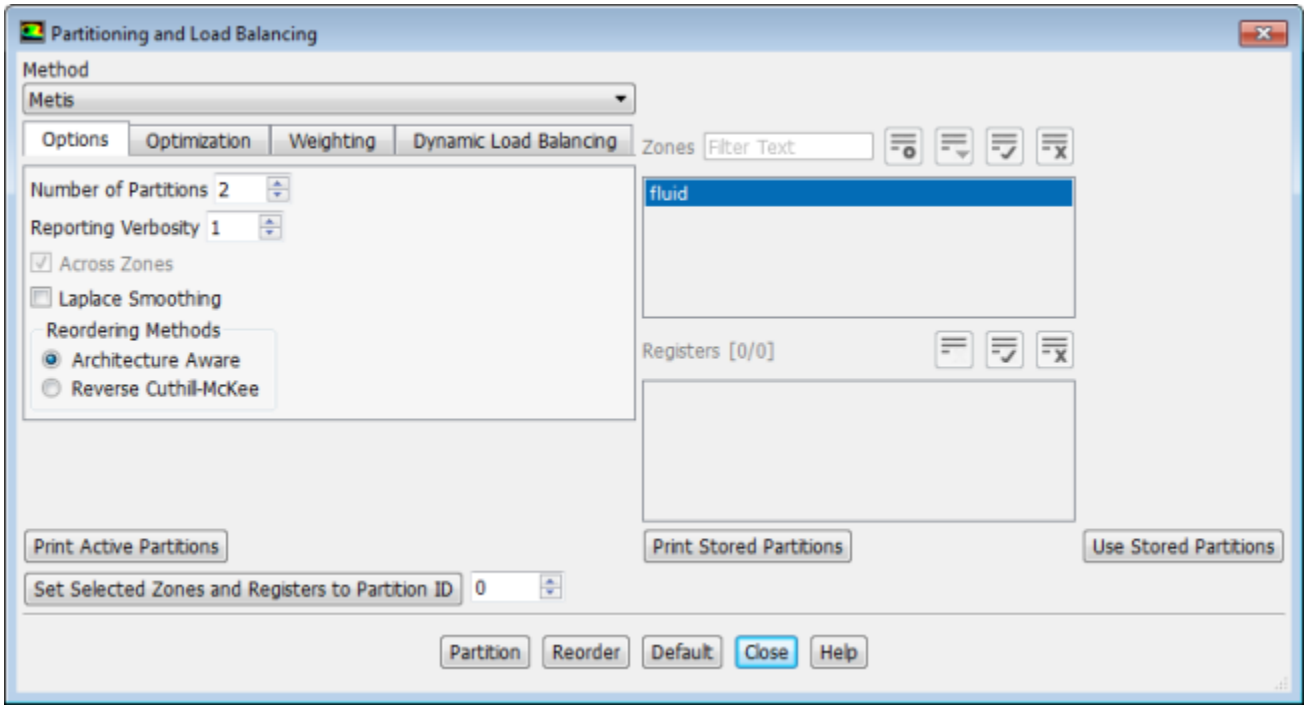
In order to partition the mesh, you must select the partition method for creating the mesh partitions, set the number of partitions, select the zones and/or registers, and choose the optimizations to be used. For some methods, you can also perform pretesting to ensure that the best possible partition is performed. Once you have set all the parameters in the **Partitioning and Load Balancing** dialog box to your satisfaction, click the **Partition** button to subdivide the mesh into the selected number of partitions using the prescribed method and optimization(s). For recommended partitioning strategies see Guidelines for Partitioning the Mesh (p. 2033).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can set the relevant inputs in the **Partitioning and Load Balancing** dialog box (Figure 34.8: The Partitioning and Load Balancing Dialog Box in the Parallel Solver (p. 2034) in the parallel solver, or Figure 34.9: The Partitioning and Load Balancing Dialog Box in the Serial Solver (p. 2035) in the serial solver) in the following manner:

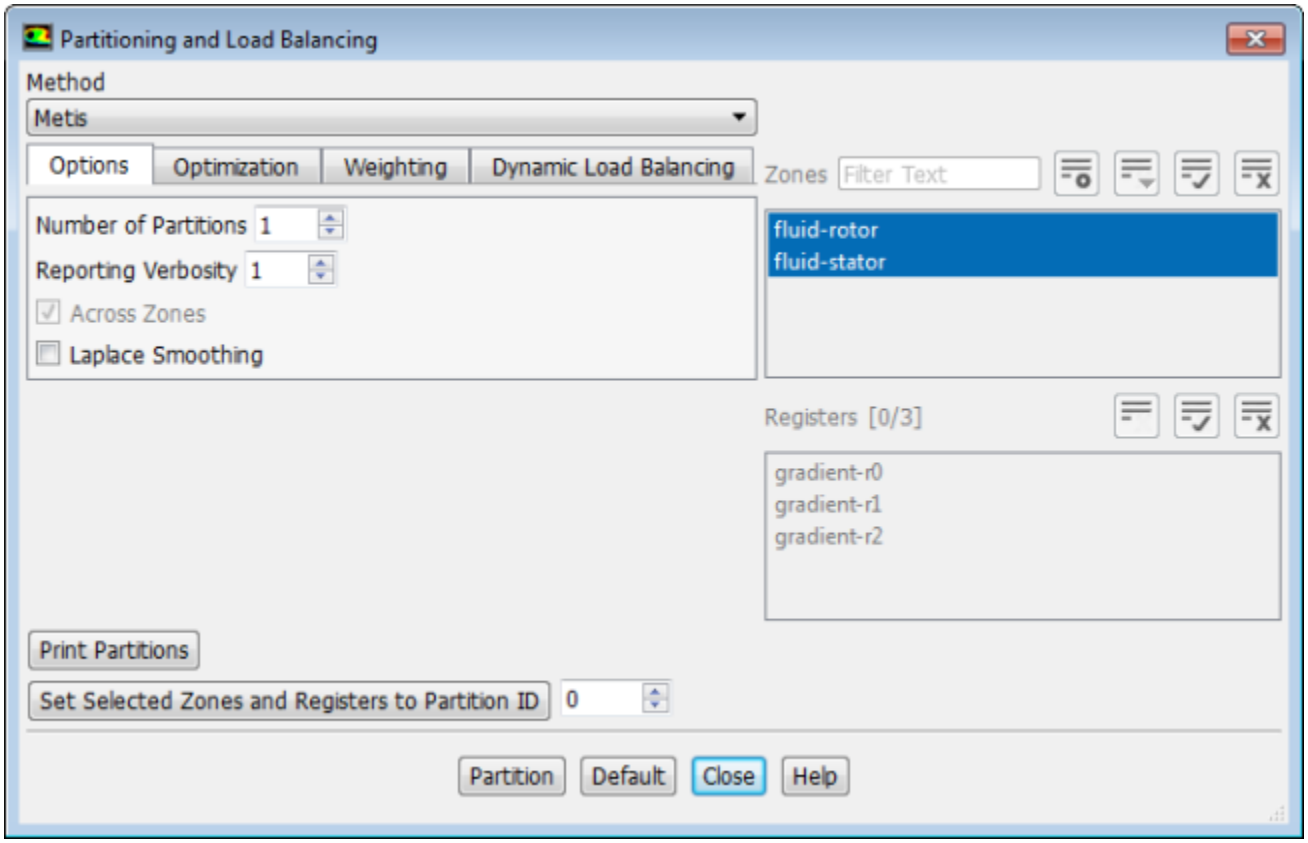
Parallel → General → Partition/Load Balance...

Figure 34.8: The Partitioning and Load Balancing Dialog Box in the Parallel Solver



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 34.9: The Partitioning and Load Balancing Dialog Box in the Serial Solver



1. Select the **Method** from the drop-down list. The choices are described in Partition Methods (p. 2046).

2. In the **Options** tab

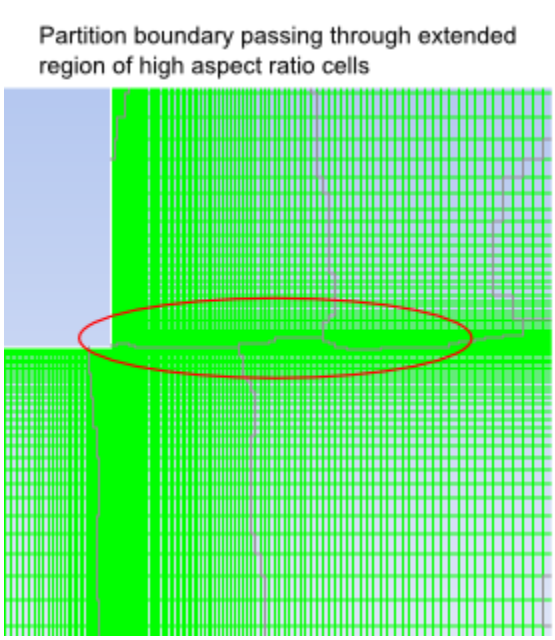
   a. Set the desired number of mesh partitions in the **Number of Partitions** field. You can use the counter arrows to increase or decrease the value, instead of typing in the box. The number of mesh partitions must be an integer number that is divisible by the number of processors available for parallel computing.

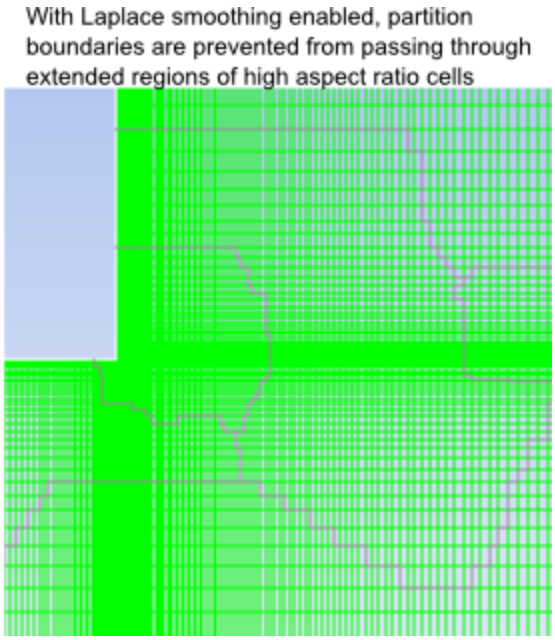
   b. Set the **Reporting Verbosity**. This allows you to control what is displayed in the console. For details, see Reporting During Partitioning (p. 2043).

   c. By default, partitioning will be done across cell zones. You can choose to independently apply partitioning to each cell zone, by disabling **Across Zones**. This is not recommended unless cells in different zones will require significantly different amounts of computation during the solution phase (for example, if the domain contains both solid and fluid zones).

   d. If you are using the **Metis** method, you have the option of enabling **Laplace Smoothing**. This option can be used to prevent partition boundaries from passing through areas of high aspect ratio cells. This can improve convergence in dynamic mesh cases and other cases with regions of highly stretched cells. After enabling Laplace Smoothing, you can specify the **Cutoff Aspect Ratio**. The Cutoff Aspect Ratio corresponds roughly to the maximum aspect ratio allowable along a partition boundary.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





e. Select the **Reordering Method** for partitions to optimize parallel performance:

- **Architecture Aware**: This is the default option and it accounts for the system architecture and network topology in remapping the partitions to the processors.
- **Reverse Cuthill-McKee**: This option minimizes the bandwidth of the compute-node connectivity matrix (the maximum distance between two connected processes) without incorporating the system architecture.

The reordering methods are parallel performance tuning options. After the case is initially partitioned for parallel processing, the partition reordering step will remap the partitions in a more optimal way to improve parallel performance.

Important

The Architecture-aware reordering method is not applicable when only a single machine is used for the simulation.

After initially loading the case into a parallel session, you can click the **Reorder** button to reorder the partitions. The necessary algorithms are executed and ANSYS Fluent will report if it can find a more optimal mapping for the partitions, as well as the potential improvement in inter-machine communications. If the reported improvement is significant (say, more than 5%), then you can click the **Use Stored Partitions** button to use the new partition mapping. This will generally entail large data transfers amongst all the processes, and another reliable method to use the new partitions would be to write out a case file and load it back in to a new parallel session. The process is similar to re-partitioning with a new partitioning method, for example. Note that sometimes, depending on the cluster configuration and initial case partitioning, and if the partitions have already been reordered, no improvement is possible, and this will be reported in the console after clicking the **Reorder** button. You can simply continue in this case, and there will be no effect on the simulation. Also, note that partition reordering is specific to the current parallel configuration and should be repeated if the number of machines used changes during subsequent computations.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. In the **Optimization** tab

   a. You can enable and control the desired optimization methods (described in Optimizations (p. 2050)). You can enable the **Merge** and **Smooth** schemes by enabling the check button next to each one. For each scheme, you can also set the number of **Iterations**. Each optimization scheme will be applied until appropriate criteria are met, or the maximum number of iterations has been executed. If the **Iterations** counter is set to 0, the optimization scheme will be applied until completion, with no limit on maximum number of iterations.

   b. Choosing the **Principal Axes** or **Cartesian Axes** method, you can improve the partitioning by enabling the automatic testing of the different bisection directions before the actual partitioning occurs. To use pretesting, enable the **Pre-Test** option. Pretesting is described in Pretesting (p. 2051).

4. In the **Zones** and/or **Registers** lists, select the zone(s) and/or register(s) for which you want to partition. For most cases, you will select all **Zones** (the default) to partition the entire domain. See below for details.

5. You can assign selected **Zones** and/or **Registers** to a specific partition ID by entering a value for the **Set Selected Zones and Registers to Partition ID**. For example, if the **Number** of partitions for your mesh is 2, then you can only use IDs of 0 or 1. If you have three partitions, then you can enter IDs of 0, 1, or 2. This can be useful in situations where the gradient at a region is known to be high. In such cases, you can mark the region or zone and set the marked cells to one of the partition IDs, thereby preventing the partition from going through that region. This in turn will facilitate convergence. This is also useful in cases where mesh manipulation tools are not available in parallel. In this case, you can assign the related cells to a particular ID so that the mesh manipulation tools are now functional.

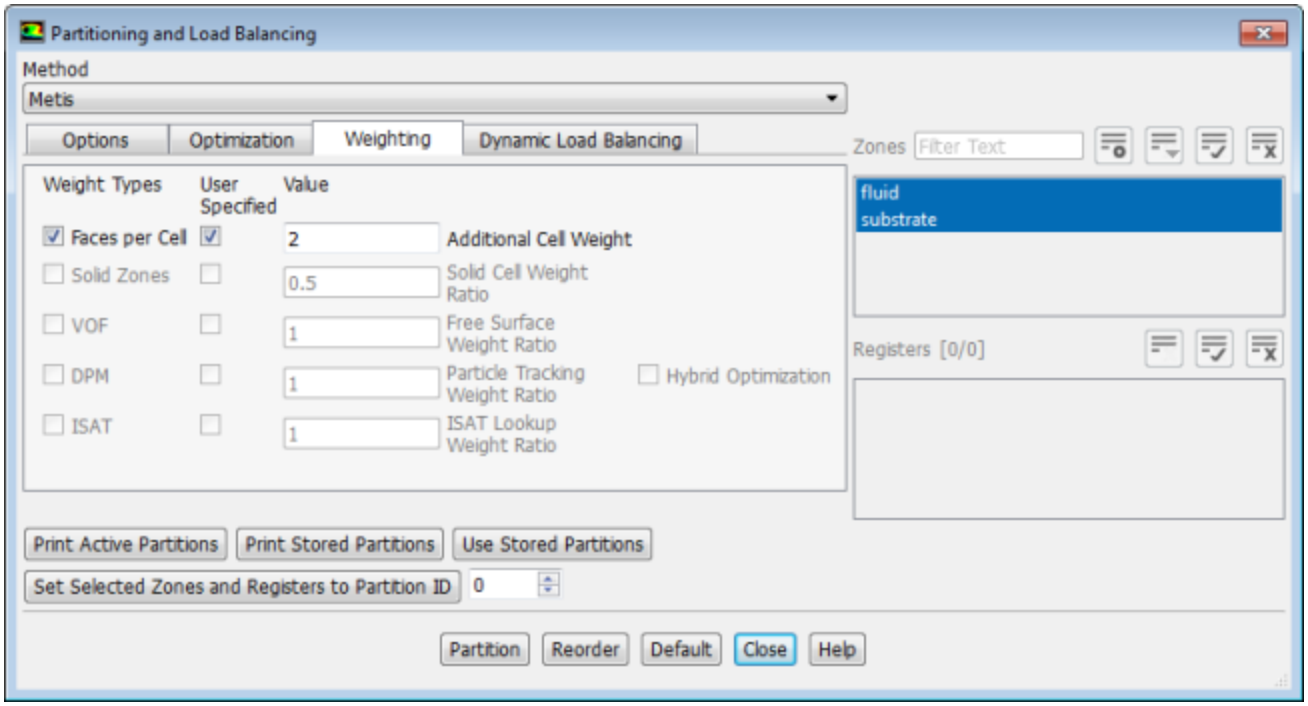
   If you are running the parallel solver, and you have marked your region and assigned an ID to the selected **Zones** and/or **Registers**, click the **Use Stored Partitions** button to make the new partitions valid.

   Refer to the example described later in this section for a demonstration of how selected registers are assigned to a partition (Example of Setting Selected Registers to Specified Partition IDs (p. 2040)).

6. In the **Weighting** tab (Figure 34.10: The Weighting Tab in the Partitioning and Load Balancing Dialog Box for the Parallel Solver (p. 2038)), you can set the appropriate weights prior to partitioning the mesh, to improve load balancing and overall performance. You can control weights for cells, solid zones, VOF, DPM, and ISAT table lookup. You can either rely on ANSYS Fluent timers to set the weight scaling, specify the value by enabling **User-Specified**, or use model-weighted partitioning (so that Fluent automatically calculates the weighting based on the cell count and the models and attributes used as weights).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 34.10: The Weighting Tab in the Partitioning and Load Balancing Dialog Box for the Parallel Solver



a. Enable **Faces per Cell** so that the partitioning assigns a weight to each cell based on its number of faces. This type of weighting is advantageous when the case has mixed or polyhedral cell zones. If you enable the **User-Specified** check box, the weight assigned to each cell will be the number of faces plus the **Additional Cell Weight** you enter in the number-entry box under **Value**. By default the **Faces per Cell** weighting is enabled with the **Additional Cell Weight** set to **2**.

b. Enable **Solid Zones** weighting to allow the partitioning to take solid zones into consideration. By default, ANSYS Fluent will scale the weight based on the computational time of the solid and fluid zones. If an iteration has not been run, then you may need to specify a value after enabling the user-specified check box. The value you enter is relative to the fluid weight. Typically, it should be less than `1` (for example, `0.0001`) so that the calculation will be quicker for the solid zone compared to the fluid zone. Entering a value greater than `1` for the **Solid Zones** means that the calculation will take longer and will be more computationally expensive for the solid zone compared to the fluid zone.

Important

Setting the **Solid Zones** either too high or too low can cause a load imbalance, depending on which equations are being solved. If you are not sure what would be an appropriate value, have ANSYS Fluent automatically set it for you by making sure that **User-Specified** is disabled.

---

c. Enable **VOF** weighting to allow the partitioning to consider the imbalance caused by the free surface reconstruction with the geo-reconstruct scheme. Therefore, it is only available when using the VOF model with geometric reconstruction. You may use the user-specified value before timers are collected, or if you want to specify a value other than timing statistics. The specified value is the VOF proportion of the total computational effort.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

d. Enable **DPM** weighting to set the weight of DPM particles relative to the continuous phase. DPM weights are valid when you have particle tracking in your simulation, where the user-specified value is the DPM proportion of the total computational effort relative to the continuous phase. Note that this is available only when you have injections defined. For details, see Modeling Discrete Phase (p. 1235).

   The DPM weight takes into account the distribution of the tracking effort over the partitions and it is available after at least one calculation step with particle tracking. Displaying Particle Tracks does not change the weights. The computational effort is determined by the number of DPM steps performed in each cell. This weight becomes more important when the time for the particle tracking of particles exceeds the time for solving the flow. Enabling this option in the **Weighting** tab enables the counting of the particle steps in the cells. These values are available for contour and vector plots when using the **Discrete Phase Model** and **DPM Steps per Cell** variable. After repartitioning, the DPM weights are reset before the next particle tracking. It is generally preferable to partition along the dominant path of the particles in order to minimize particles crossing partition boundaries and thereby reducing associated communication costs. However, partitioning should also consider load balance for the other models, especially the continuous phase, and model weighting provides a means to effectively load balance the overall simulation.

   Select the **Hybrid Optimization** option to enable the hybrid optimization partition weighting method for DPM. This method balances the load across machines, and, within each machine, the hybrid parallel DPM method is used to make sure the load is balanced by multi-threading. First, the domain is split based on the model weights of each cell and then partitioned across a number of machines. Finally, each machine is partitioned according to the number of cores. This allows you to have a balanced number of cells in each partition, at the same time having a balanced number of particles on each machine, which will be further balanced by the hybrid DPM method. This optimization option is also applicable to the discrete element method (DEM) collision model.

e. Enable **ISAT** weighting to balance the load during the ISAT table lookup for the stiff-chemistry Laminar, EDC or PDF Transport models. The ISAT algorithm builds an unstructured table in N species dimensions for storage and retrieval of the chemistry mappings. Since chemistry is usually computationally expensive, this storage/retrieval can be very time-consuming (for information about ISAT, refer to In-Situ Adaptive Tabulation (ISAT) in the Theory Guide). Each parallel node builds its own table, and there is no message passing to tables on other nodes. As some nodes may have more chemical reactions than others (for example one parallel node may contain just air at a constant temperature, in which case the ISAT table will contain only one entry and calculation will be rapid), there may be a load imbalance. The dynamic load balancing algorithm will migrate cells from high computational load nodes to low computational load nodes.

   If you decide to specify a value, this user-specified value is the ISAT proportion of the total computational effort.

f. You have the option of using model-weighted partitioning. The objective of model-weighted partitioning is to balance the overall number of cells, as well as the time needed for the selected models (that is, the enabled **Weight Types**, described previously). This is specifically useful for cases with models that could potentially lead to a load imbalance—for example, when using the discrete phase model, the distribution of particles could be different across partitions and may cause an imbalance, or in cases with large proportion of solid zones.

   Each model is considered as a constraint, and a base constraint is automatically introduced for the overall number of cells. When partitioning, ANSYS Fluent will automatically calculate the weights for these constraints, and balance each of them.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To enable model-weighted partitioning, ensure that the **Metis** partition method is selected, and enable the appropriate models under **Weight Types** in the **Weighting** tab. Then use the following text command prior to partitioning:

`parallel → partition → set → model-weighted-partition`

Note that you can get additional information specific to the constraints by setting the **Reporting Verbosity** to 2 in the **Options** tab.

7. When using the dynamic mesh model in your parallel simulations, the **Partition** dialog box includes an **Auto Repartition** option and a **Repartition Interval** setting. These parallel partitioning options are provided because ANSYS Fluent migrates cells when local remeshing and smoothing are performed. Therefore, the partition interface becomes very wrinkled and the load balance may deteriorate. By default, the **Auto Repartition** option is selected, where a percentage of interface faces and loads are automatically traced. When this option is selected, ANSYS Fluent automatically determines the most appropriate repartition interval based on various simulation parameters. Sometimes, using the **Auto Repartition** option provides insufficient results, therefore, the **Repartition Interval** setting can be used. The **Repartition Interval** setting lets you to specify the interval (in time steps or iterations respectively) when a repartition is enforced. When repartitioning is not desired, you can set the **Repartition Interval** to zero.

Important

Note that when dynamic meshes and local remeshing is utilized, updated meshes may be slightly different in parallel ANSYS Fluent (when compared to serial ANSYS Fluent or when compared to a parallel solution created with a different number of compute nodes), resulting in very small differences in the solutions.

---

8. Click the **Partition** button to partition the mesh.

9. If you decide that the new partitions are better than the previous ones (if the mesh was already partitioned), click the **Use Stored Partitions** button to make the newly stored cell partitions the active cell partitions. The active cell partition is used for the current calculation, while the stored cell partition (the last partition performed) is used when you save a case file.

34.5.4.1.1. Example of Setting Selected Registers to Specified Partition IDs

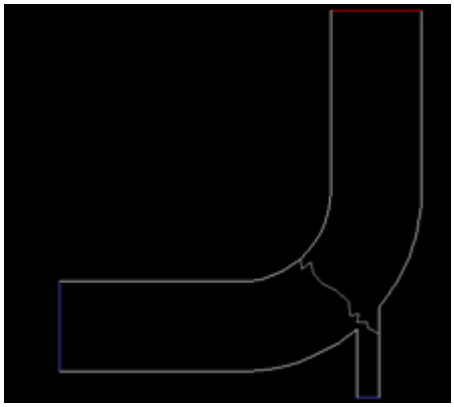
1. Start ANSYS Fluent in parallel. The case in this example was partitioned across two nodes.

2. Read in your case.

3. Display the mesh with the **Partitions** option enabled in the **Mesh Display** dialog box (Figure 34.11: The Partitioned Mesh (p. 2041)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 34.11: The Partitioned Mesh



4. Adapt your region and mark your cells (for details, see Performing Region Adaption (p. 1693)). This creates a register.

5. Open the **Partitioning and Load Balancing** dialog box.

6. Set the **Set Selected Zones and Registers to Partition ID** to `0` and click the corresponding button. This displays the following output in the ANSYS Fluent console:

```
>> 2 Active Partitions:
  ------------------------------------------------------------------------
  Collective Partition Statistics:      Minimum      Maximum   Total
  ------------------------------------------------------------------------
  Cell count                            459          459       918
  Mean cell count deviation             0.0%         0.0%
  Partition boundary cell count         11           11        22
  Partition boundary cell count ratio   2.4%         2.4%      2.4%

  Face count                            764         1714       2461
  Mean face count deviation             -38.3%       38.3%
  Partition boundary face count         13          13        17
  Partition boundary face count ratio   0.8%        1.7%      0.7%

  Partition neighbor count              1           1
  ------------------------------------------------------------------------
  Partition Method                      Metis
  Stored Partition Count                2
Done.
```

7. Click the **Use Stored Partitions** button to make the new partitions valid. This migrates the partitions to the compute-nodes. The following output is then displayed in the ANSYS Fluent console:

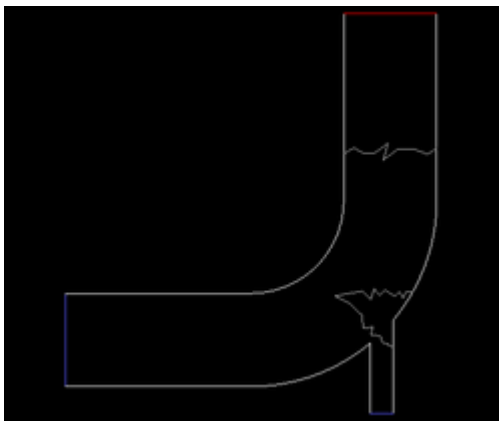
```
Migrating partitions to compute-nodes.
>> 2 Active Partitions:
     P   Cells  I-Cells  Cell Ratio   Faces  I-Faces  Face Ratio  Neighbors
     0     672      24        0.036    2085       29      0.014          1
     1     246      24        0.098     425       29      0.068          1
  ------------------------------------------------------------------------
  Collective Partition Statistics:     Minimum       Maximum      Total
  ------------------------------------------------------------------------
  Cell count                           246           672          918
  Mean cell count deviation            -46.4%        46.4%
  Partition boundary cell count        24            24           48
  Partition boundary cell count ratio  3.6%          9.8%         5.2%

  Face count                           425           2085         2461
  Mean face count deviation            -66.1%        66.1%
  Partition boundary face count        29            29           49
```

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
  Partition boundary face count ratio  1.4%          6.8%         2.0%

  Partition neighbor count             1             1
  ------------------------------------------------------------------------
  Partition Method                     Metis
  Stored Partition Count               2
Done.
```

8. Display the mesh (Figure 34.12: The Partitioned ID Set to Zero (p. 2042)).

Figure 34.12: The Partitioned ID Set to Zero

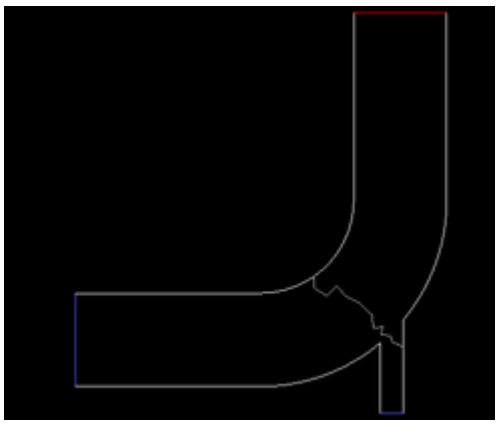


9. This time, set the **Set Selected Zones and Registers to Partition ID** to 1 and click the corresponding button. This displays a report in the ANSYS Fluent console.

10. Click the **Use Stored Partitions** button to make the new partitions valid and to migrate the partitions to the compute-nodes.

11. Display the mesh (Figure 34.13: The Partitioned ID Set to 1 (p. 2042)). Notice now that the partition appears in a different location as specified by your partition ID.

Figure 34.13: The Partitioned ID Set to 1



Important

Although this example demonstrates setting selected registers to specific partition IDs in parallel, it can be similarly applied in serial.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

34.5.4.1.2. Partitioning Within Zones or Registers

The ability to restrict partitioning to cell zones or registers gives you the flexibility to apply different partitioning strategies to subregions of a domain. For example, if your geometry consists of a cylindrical plenum connected to a rectangular duct, you may want to partition the plenum using the **Cylindrical Axes** method, and the duct using the **Cartesian Axes** method.

If the plenum and the duct are contained in two different cell zones, you can select one at a time and perform the desired partitioning, as described in Using the Partitioning and Load Balancing Dialog Box (p. 2033). If they are not in two different cell zones, you can create a cell register (basically a list of cells) for each region using the functions that are used to mark cells for adaption. These functions allow you to mark cells based on physical location, cell volume, gradient or isovalue of a particular variable, and other parameters. See Adapting the Mesh (p. 1681) for information about marking cells for adaption. Manipulating Adaption Registers (p. 1701) provides information about manipulating different registers to create new ones. Once you have created a register, you can partition within it as described in Example of Setting Selected Registers to Specified Partition IDs (p. 2040).

Important

Note that partitioning within zones or registers is not available when **Metis** is selected as the partition **Method**.

For dynamic mesh applications, ANSYS Fluent stores the partition method used to partition the respective zone. Therefore, if repartitioning is done, ANSYS Fluent uses the same method that was used to partition the mesh.

34.5.4.1.3. Reporting During Partitioning

As the mesh is partitioned, information about the partitioning process will be displayed in the console. By default, the number of partitions created, the time required for the partitioning, and the minimum and maximum cell, face, interface, and face-ratio variations will be displayed (for details, see Interpreting Partition Statistics (p. 2052)). If you increase the **Reporting Verbosity** to 2 from the default value of 1, the partition method used, the partition ID, number of cells, faces, and interfaces, and the ratio of interfaces to faces for each partition will also be displayed in the console. If you decrease the **Reporting Verbosity** to 0, only the number of partitions created and the time required for the partitioning will be reported.

You can request a portion of this report to be displayed again after the partitioning is completed. When you click the **Print Active Partitions** or **Print Stored Partitions** button in the parallel solver, ANSYS Fluent will display the partition ID, number of cells, faces, and interfaces, and the ratio of interfaces to faces for each active or stored partition in the console. In addition, it will display the minimum and maximum cell, face, interface, and face-ratio variations. In the serial solver, you will obtain the same information about the stored partition when you click **Print Partitions**. For details, see Interpreting Partition Statistics (p. 2052).

Important

Recall that to make the stored cell partitions the active cell partitions you must click the **Use Stored Partitions** button. The active cell partition is used for the current calculation, while the stored cell partition (the last partition performed) is used when you save a case file.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

34.5.4.1.4. Resetting the Partition Parameters

If you change your mind about your partition parameter settings, you can easily return to the default settings assigned by ANSYS Fluent by clicking on the **Default** button. When you click the **Default** button, it will become the **Reset** button. The **Reset** button allows you to return to the most recently saved settings (that is, the values that were set before you clicked on **Default**). After execution, the **Reset** button will become the **Default** button again.

34.5.4.2. Load Balancing

A dynamic load balancing capability is available in ANSYS Fluent. The principal reason for using parallel processing is to reduce the turnaround time of your simulation, which may be achieved by the following means:

- Faster machines, for example, faster CPU, memory, cache, and communication bandwidth between the CPU and memory
- Faster interconnects, for example, smaller latency and larger bandwidth
- Better Load balancing, for example, load is evenly distributed and CPUs are not idled during calculation

The first two evolve at the pace of computer technology, which is beyond the scope of this document. The third item is regarding optimization of available computation power. Here we are mainly talking about load balancing on dedicated homogeneous resources, which is often the case nowadays. If you are not using a dedicated homogeneous resource, you may need to account for differences in CPU speeds during partitioning by specifying a load distribution (for details, see Load Distribution (p. 2055)).

On a dedicated homogeneous system, the key for load balancing is how to evaluate the computational requirement of each cell. By default, ANSYS Fluent assumes that each cell requires the same computational work, but this is often not the case. For example

- A hexahedral cell demands more CPU and memory than a tetrahedral cell.
- A cell with particle tracking will use more time than a cell without particle tracking.
- ISAT species model cells may have magnitude differences in time usage.

To balance these differences, ideally, the time used in each cell could be recorded and load balance achieved based on these detailed timing statistics. However, this can be expensive and such low level timings can be unreliable in any case. Instead, we identify features causing computational imbalance and record time usage for these models in aggregate. For a more detailed description of this, refer to Partitioning (p. 2033) in the discussion of the **Weighting** tab. In addition, the imbalance may happen dynamically during run time, for example

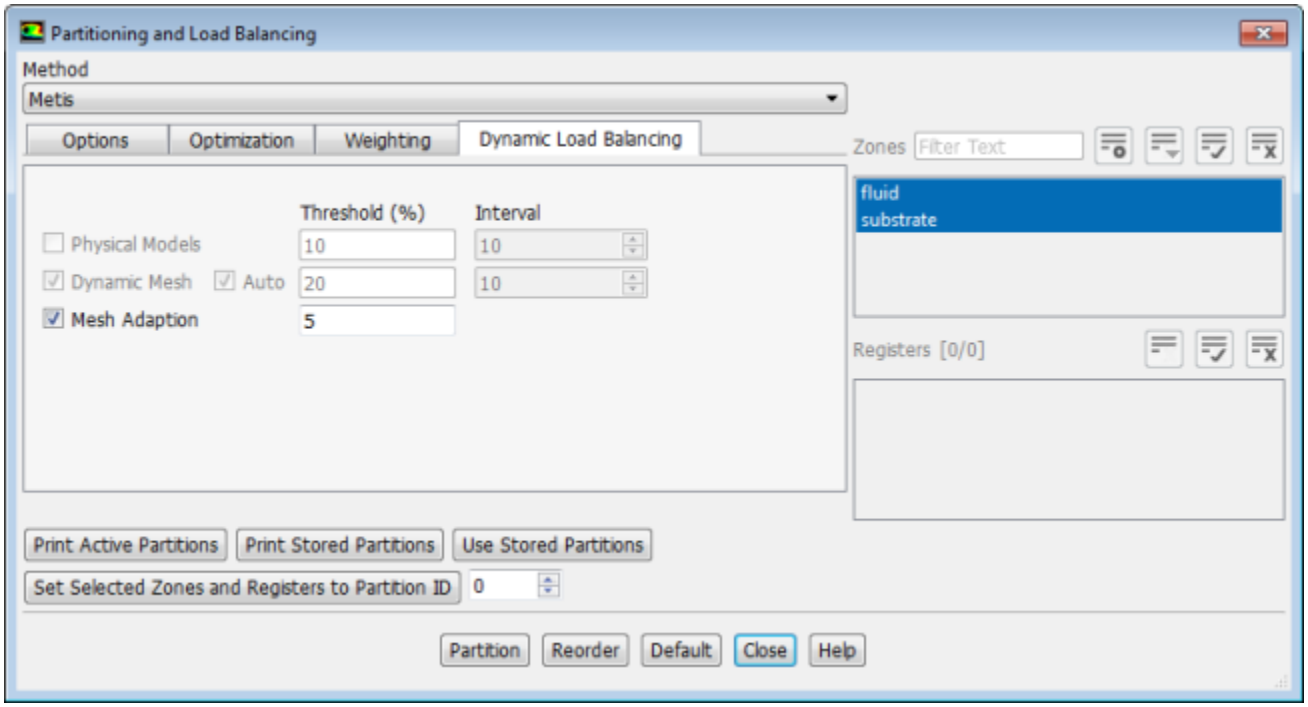
- The mesh may be changed by adaption or mesh movement.
- In unsteady cases, particle tracking may move from one region to another region.

Dynamic load balancing has been implemented for better scalability of cases with imbalanced physical or geometrical models, thereby reducing the simulation time. The implementation considers weights from these models scaled by CPU time usage. Load balancing for DPM, VOF, cell type (number of faces per cell), and solid zones can be performed. In addition, cell weight based load balancing and machine load distribution can also be specified (for details, see Load Distribution (p. 2055)). ANSYS Fluent takes the weights from physical models and considers them for partitioning. The weights are assembled based

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

on the time used by each physical model. For dynamic load balancing, the load is checked and balanced based on your specified imbalance threshold. To apply dynamic load balancing on the various models, click the **Dynamic Load Balancing** tab and select the required balancing as follows:

Figure 34.14: The Dynamic Load Balancing Tab



1. Enable **Physical Models** load balancing during iterations so that the load will be evaluated for time usage and weight distribution, based on the **Interval** that you provide. If the imbalance exceeds the specified **Threshold**, then repartitioning will be performed by considering the selected weights. **Physical Models** load balancing will only be available when you have the specific physical models enabled in the case. You will be prompted to enable the weights for those models. When weights for the physical models are all disabled, you will be prompted to disable **Physical Models** load balancing.

   Note

   Applying load balancing too frequently may cause performance degradation due to the additional cost of migrating cells for the new partition layout.

2. Enable **Dynamic Mesh** if there is any dynamic mesh movement. Load balancing, based on the number of cells, will be checked and balanced if the imbalance threshold is exceeded. These parallel partitioning options are provided because with mesh motion, when local remeshing and smoothing are performed, the partition interface can become very wrinkled and load balance may deteriorate. By default, the **Auto** option is selected, where a percentage of interface faces and loads are automatically traced. When this option is selected, ANSYS Fluent automatically determines the most appropriate repartitioning interval based on various simulation parameters. However, sometimes, the frequency of load balancing from the **Auto** option may be inadequate, and then the **Interval** setting can be explicitly set. The **Interval** setting lets you specify the interval (in time steps or iterations, respectively) when load balancing is enforced. When load balancing is

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

not desired, you may disable **Dynamic Mesh** load balancing. Dynamic Mesh load balancing is only available when you have dynamic models enabled in your case.

Important

Note that when dynamic meshes and local remeshing are utilized, updated meshes may be slightly different in parallel ANSYS Fluent (when compared to serial ANSYS Fluent or when compared to a parallel solution created with a different number of compute nodes), resulting in very small differences in the solutions.

---

3. Enable **Mesh Adaption**. Any time mesh adaption occurs, load balancing, based on the number of cells, will be checked and balanced if the imbalance threshold is exceeded. If problems arise in your computations due to adaption, you can disable the load balancing for **Mesh Adaption**.

34.5.5. Mesh Partitioning Methods

Partitioning the mesh for parallel processing has three major goals:

- Create partitions with equal numbers of cells.
- Minimize the number of partition interfaces — that is, decrease partition boundary surface area.
- Minimize the number of partition neighbors.

Balancing the partitions (equalizing the number of cells) ensures that each processor has an equal load and that the partitions will be ready to communicate at about the same time. Since communication between partitions can be a relatively time-consuming process, minimizing the number of interfaces can reduce the time associated with this data interchange. Minimizing the number of partition neighbors reduces the chances for network and routing contentions. In addition, minimizing partition neighbors is important on machines where the cost of initiating message passing is expensive compared to the cost of sending longer messages. This is especially true for workstations connected in a network.

The partitioning schemes in ANSYS Fluent use bisection or METIS algorithms to create the partitions, but unlike other schemes that require the number of partitions to be a factor of two, these schemes have no limitations on the number of partitions. You will create as many partitions as there are computing units (cores based on processors and machines) available for your simulation.

34.5.5.1. Partition Methods

The mesh is partitioned using a bisection or METIS algorithm. The selected algorithm is applied to the parent domain, and then recursively applied to the subdomains. For example, to divide the mesh into four partitions with a bisection method, Fluent will bisect the entire (parent) domain into two child domains, and then repeat the bisection for each of the child domains, yielding four partitions in total. To divide the mesh into three partitions with a bisection method, Fluent will "bisect" the parent domain to create two partitions—one approximately twice as large as the other—and then bisect the larger child domain again to create three partitions in total. METIS uses graph partitioning techniques that generally provide more optimal partitions than the geometric methods.

The mesh can be partitioned using one of the algorithms listed below. The most efficient choice is problem-dependent, so you can try different methods until you find the one that is best for your problem. See Guidelines for Partitioning the Mesh (p. 2033) for recommended partitioning strategies.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cartesian Axes
bisects the domain based on the Cartesian coordinates of the cells (see Figure 34.15: Partitions Created with the Cartesian Axes Method (p. 2048)). It bisects the parent domain and all subsequent child subdomains perpendicular to the coordinate direction with the longest extent of the active domain. It is often referred to as coordinate bisection.

Cartesian Strip
uses coordinate bisection but restricts all bisections to the Cartesian direction of longest extent of the parent domain (see Figure 34.16: Partitions Created with the Cartesian Strip or Cartesian X-Coordinate Method (p. 2049)). You can often minimize the number of partition neighbors using this approach.

Cartesian X-, Y-, Z-Coordinate
bisects the domain based on the selected Cartesian coordinate. It bisects the parent domain and all subsequent child subdomains perpendicular to the specified coordinate direction. (See Figure 34.16: Partitions Created with the Cartesian Strip or Cartesian X-Coordinate Method (p. 2049).)

Cartesian R Axes
bisects the domain based on the shortest radial distance from the cell centers to that Cartesian axis (x, y, or z) whichever produces the smallest interface size. This method is available only in 3D.

Cartesian RX-, RY-, RZ-Coordinate
bisects the domain based on the shortest radial distance from the cell centers to the selected Cartesian axis (x, y, or z). These methods are available only in 3D.

Cylindrical Axes
bisects the domain based on the cylindrical coordinates of the cells. This method is available only in 3D.

Cylindrical R-, Theta-, Z-Coordinate
bisects the domain based on the selected cylindrical coordinate. These methods are available only in 3D.

Metis
uses the METIS software package for partitioning irregular graphs, developed by Karypis and Kumar at the University of Minnesota and the Army HPC Research Center. It uses a multilevel approach in which the vertices and edges on the fine graph are coalesced to form a coarse graph. The coarse graph is partitioned, and then uncoarsened back to the original graph. During coarsening and uncoarsening, algorithms are applied to permit high-quality partitions. Detailed information about METIS can be found in its manual [49] (p. 2889).

Important

If you create non-conformal interfaces, and generate virtual polygonal faces, your METIS partition can cross non-conformal interfaces by using the connectivity of the virtual polygonal faces. This improves load balancing for the parallel solver and minimizes communication by decreasing the number of partition interface cells.

Polar Axes
bisects the domain based on the polar coordinates of the cells (see Figure 34.19: Partitions Created with the Polar Axes or Polar Theta-Coordinate Method (p. 2050)). This method is available only in 2D.

Polar R-Coordinate, Polar Theta-Coordinate
bisects the domain based on the selected polar coordinate (see Figure 34.19: Partitions Created with the Polar Axes or Polar Theta-Coordinate Method (p. 2050)). These methods are available only in 2D.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Principal Axes

bisects the domain based on a coordinate frame aligned with the principal axes of the domain (see Figure 34.17: Partitions Created with the Principal Axes Method (p. 2049)). This reduces to Cartesian bisection when the principal axes are aligned with the Cartesian axes. The algorithm is also referred to as moment, inertial, or moment-of-inertia partitioning.

This is the default bisection method in ANSYS Fluent.

Principal Strip

uses moment bisection but restricts all bisections to the principal axis of longest extent of the parent domain (see Figure 34.18: Partitions Created with the Principal Strip or Principal X-Coordinate Method (p. 2050)). You can often minimize the number of partition neighbors using this approach.

Principal X-, Y-, Z-Coordinate

bisects the domain based on the selected principal coordinate (see Figure 34.18: Partitions Created with the Principal Strip or Principal X-Coordinate Method (p. 2050)).

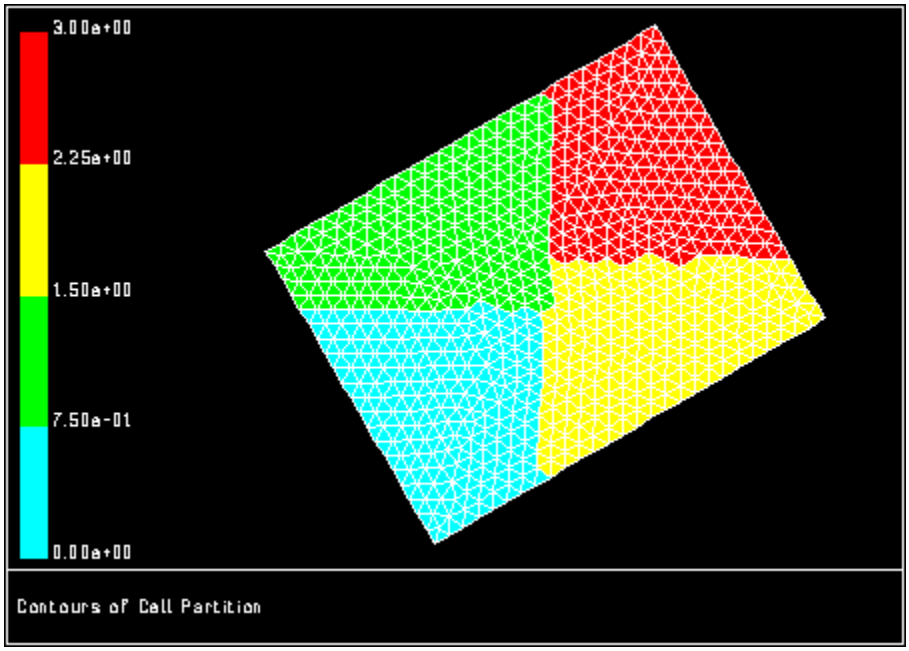
Spherical Axes

bisects the domain based on the spherical coordinates of the cells. This method is available only in 3D.

Spherical Rho-, Theta-, Phi-Coordinate

bisects the domain based on the selected spherical coordinate. These methods are available only in 3D.

Figure 34.15: Partitions Created with the Cartesian Axes Method



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 34.16: Partitions Created with the Cartesian Strip or Cartesian X-Coordinate Method

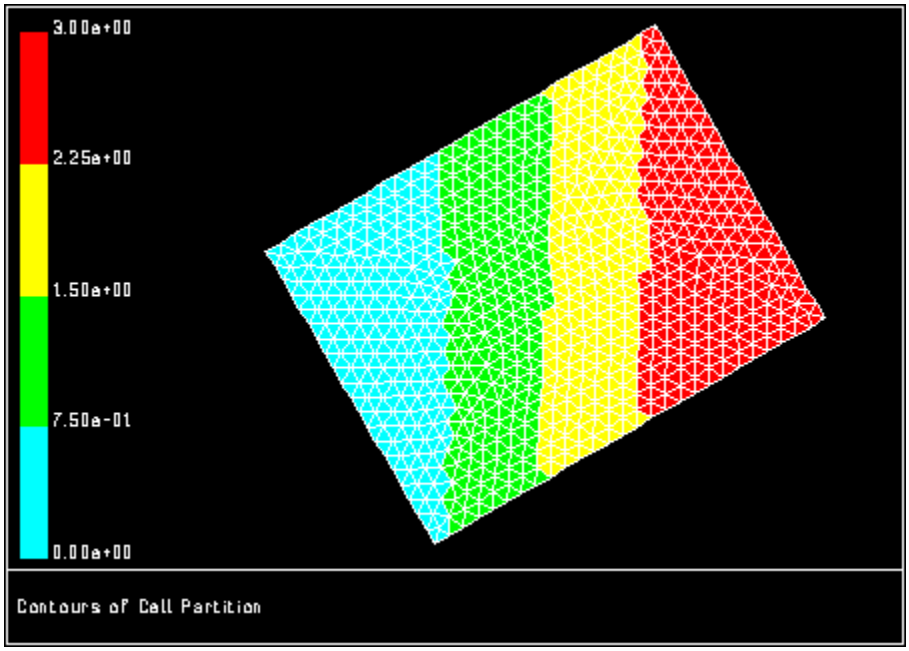
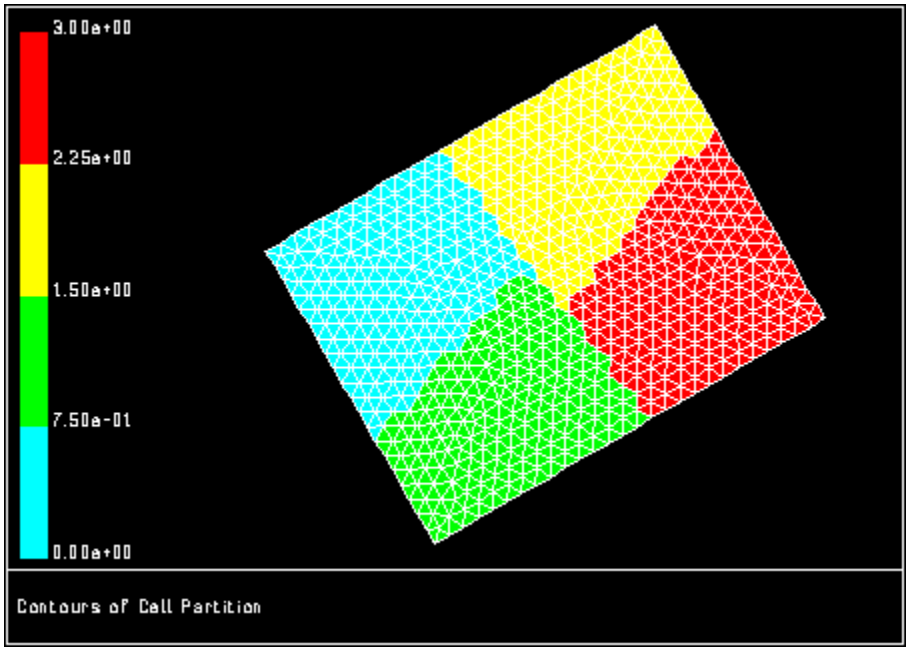


Figure 34.17: Partitions Created with the Principal Axes Method



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 34.18: Partitions Created with the Principal Strip or Principal X-Coordinate Method

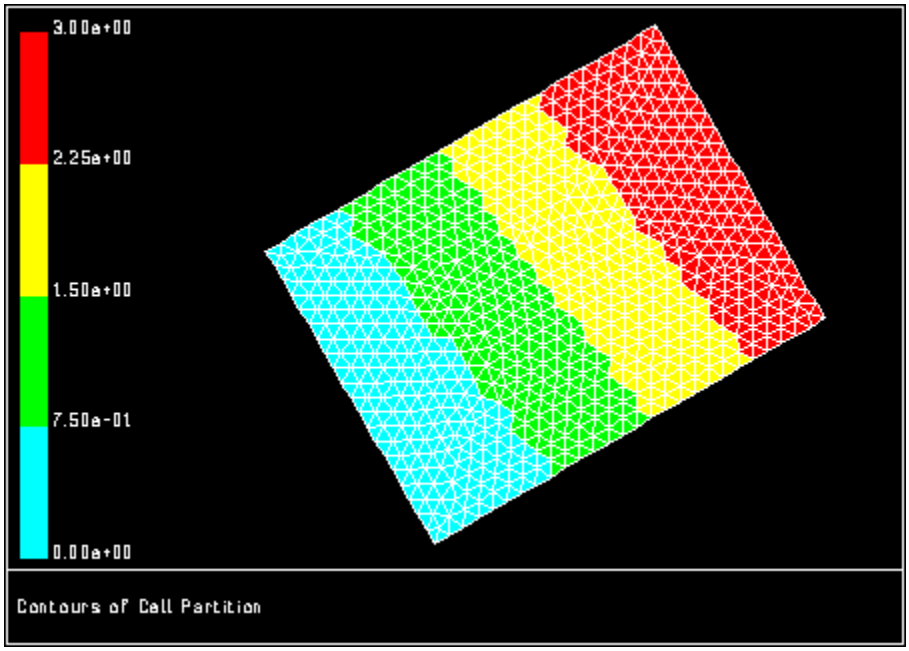
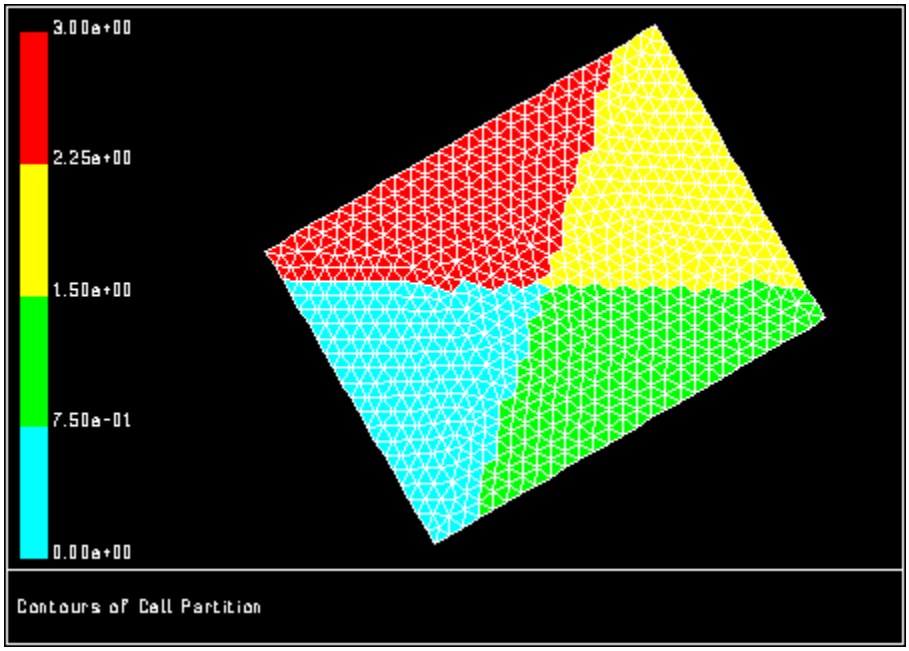


Figure 34.19: Partitions Created with the Polar Axes or Polar Theta-Coordinate Method



34.5.5.2. Optimizations

Additional optimizations can be applied to improve the quality of the mesh partitions. The heuristic of bisecting perpendicular to the direction of longest domain extent is not always the best choice for creating the smallest interface boundary. A pre-testing operation (for details, see Pretesting (p. 2051)) can be applied to automatically choose the best direction before partitioning. In addition, the following iterative optimization schemes exist:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Smooth

attempts to minimize the number of partition interfaces by swapping cells between partitions. The scheme traverses the partition boundary and gives cells to the neighboring partition if the interface boundary surface area is decreased. (See Figure 34.20: The Smooth Optimization Scheme (p. 2051).)

Merge

attempts to eliminate orphan clusters from each partition. An orphan cluster is a group of cells with the common feature that each cell within the group has at least one face that coincides with an interface boundary. (See Figure 34.21: The Merge Optimization Scheme (p. 2051).) Orphan clusters can degrade multigrid performance and lead to large communication costs.

Figure 34.20: The Smooth Optimization Scheme

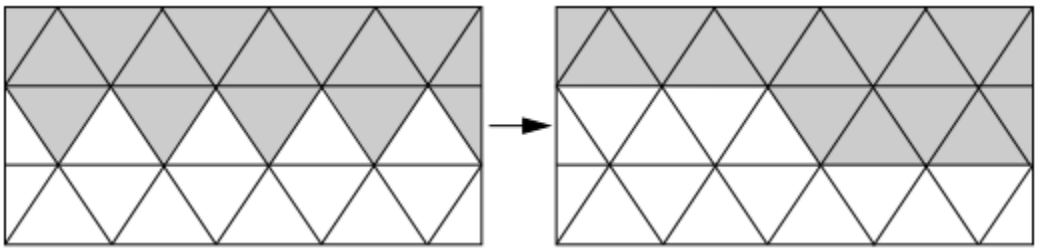
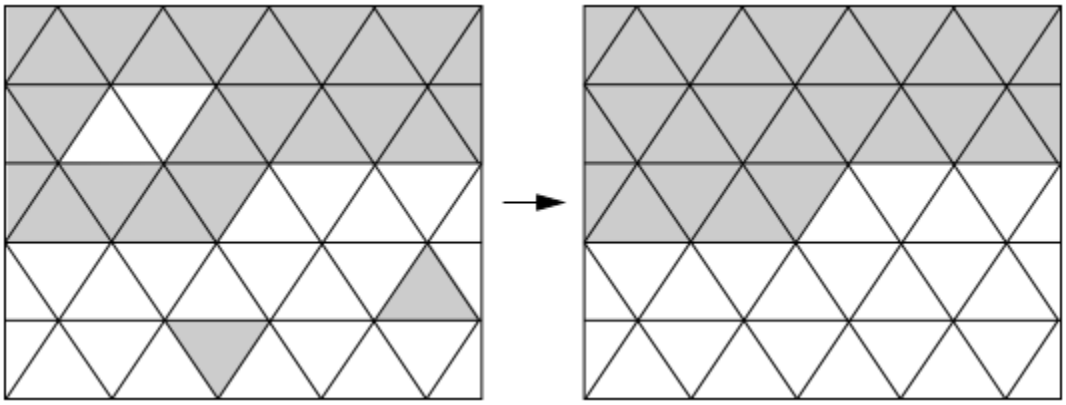


Figure 34.21: The Merge Optimization Scheme



In general, the Smooth and Merge schemes are relatively inexpensive optimization tools.

34.5.5.3. Pretesting

If you choose the **Principal Axes** or **Cartesian Axes** method, you can improve the bisection by testing different directions before performing the actual bisection. If you choose not to use pretesting (the default), ANSYS Fluent will perform the bisection perpendicular to the direction of longest domain extent.

If pretesting is enabled, it will occur automatically when you click the **Partition** button in the Partitioning and Load Balancing Dialog Box (p. 2767), or when you read in the mesh if you are using automatic partitioning. The bisection algorithm will test all coordinate directions and choose the one which yields the fewest partition interfaces for the final bisection.

Note that using pretesting will increase the time required for partitioning. For 2D problems partitioning will take 3 times longer than without pretesting, and for 3D problems it will take 4 times longer.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

34.5.5.4. Using the Partition Filter

As noted above, you can use the METIS partitioning method through a filter in addition to within the **Auto Partition Mesh** and **Partitioning and Load Balancing** dialog boxes. To perform METIS partitioning on an unpartitioned mesh, use the **File/Import/Partition/Metis...** ribbon tab item.

File → Import → Partition → Metis...

ANSYS Fluent will use the METIS partitioner to partition the mesh, and then read the partitioned mesh. The number of partitions will be equal to the number of processes. You can then proceed with the model definition and solution.

Important

Direct import to the parallel solver through the partition filter requires that the host machine has enough memory to run the filter for the specified mesh. If not, you must run the filter on a machine that does have enough memory. You can either start the parallel solver on the machine with enough memory and repeat the process described above, or run the filter manually on the new machine and then read the partitioned mesh into the parallel solver on the host machine.

To manually partition a mesh using the partition filter, enter the following command:

`utility partition` *input_filename partition_count output_filename*

where *input_filename* is the filename for the mesh to be partitioned, *partition_count* is the number of partitions desired, and *output_filename* is the filename for the partitioned mesh. You can then read the partitioned mesh into Fluent (using the standard **File/Read/Case...** ribbon tab item) and proceed with the model definition and solution.

When the **File/Import/Partition/Metis...** ribbon tab item is used to import an unpartitioned mesh into the parallel solver, the METIS partitioner partitions the entire mesh. You may also partition each cell zone individually, using the **File/Import/Partition/Metis Zone...** ribbon tab item.

File → Import → Partition → Metis Zone...

This method can be useful for balancing the work load. For example, if a case has a fluid zone and a solid zone, the computation in the fluid zone is more expensive than in the solid zone, so partitioning each zone individually will result in a more balanced work load.

34.5.6. Checking the Partitions

After partitioning a mesh, you should check the partition information and examine the partitions graphically.

34.5.6.1. Interpreting Partition Statistics

You can request a report to be displayed after partitioning (either automatic or manual) is completed. In the parallel solver, click the **Print Active Partitions** or **Print Stored Partitions** button in the **Partitioning and Load Balancing** dialog box. In the serial solver, click the **Print Partitions** button.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS Fluent distinguishes between two cell partition schemes within a parallel problem: the active cell partitions and the stored cell partitions. Initially, both are set to the cell partitions that were established upon reading the case file. If you re-partition the mesh using the **Partitioning and Load Balancing** dialog box, the new partitions will be referred to as the stored cell partitions. To make them the active cell partitions, you must click the **Use Stored Partitions** button in the **Partitioning and Load Balancing** dialog box. The active cell partitions are used for the current calculation, while the stored cell partitions (determined from the last partitioning performed) are used when you save a case file. This distinction is made mainly to allow you to partition a case on one machine or network of machines and solve it on a different one. Thanks to the two separate partitioning schemes, you could use the parallel solver with a certain number of compute nodes to subdivide a mesh into an arbitrary different number of partitions, suitable for a different parallel machine, save the case file, and then load it into the designated machine.

When you click **Print Partitions** in the serial solver, you will obtain information about the stored partitions.

The output generated when you print the partitions consists of tabulated information about the active or stored partitioning scheme. A typical output for a mesh with 4 partitions is as follows:

```
 >> 4 Active Partitions:
     P   Cells I-Cells Cell Ratio   Faces I-Faces Face Ratio Neighbors Load
     0    3520     142      0.040   11399     195      0.017         1    1
     1    3298     115      0.035   10678     151      0.014         1    1
     2    3451     305      0.088   11404     372      0.033         2    1
     3    3583     332      0.093   11586     416      0.036         2    1

    ------------------------------------------------------------------------
    Collective Partition Statistics:        Minimum   Maximum   Total
    ------------------------------------------------------------------------
    Cell count                              3298      3583      13852
    Mean cell count deviation               -4.8%     3.5%
    Partition boundary cell count           115       332       894
    Partition boundary cell count ratio     3.5%      9.3%      6.5%

    Face count                              10678     11586     44500
    Mean face count deviation               -5.2%     2.8%
    Partition boundary face count           151       416       567
    Partition boundary face count ratio     1.4%      3.6%      1.3%

    Partition neighbor count                1         2
    ------------------------------------------------------------------------
    Partition Method                        Metis
    Stored Partition Count                  4
```

The first table in the output displays per-partition statistics of interest:

P
the partition ID

Cells
the number of cells in the partition

I-Cells
the number of interface cells in the partition (that is, cells that lie on the partition interfaces)

Cell Ratio
the ratio of interface cells to total cells for the partition

Faces
the number of faces in the partition

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

I-Faces
the number of interface faces in the partition (that is, faces that lie on partition interfaces)

Face Ratio
the ratio of interface faces to total faces for the partition

Neighbors
the number of neighbor partitions

Load
the desired relative load on this node in proportion to the other nodes. See Load Distribution (p. 2055) for details.

Note that partition IDs correspond directly to compute node IDs when a case file is read into the parallel solver. When the number of partitions in a case file is larger than the number of compute nodes, but is evenly divisible by the number of compute nodes, then the distribution is such that partitions with IDs 0 to $\left(M-1\right)$ are mapped onto compute node 0, partitions with IDs M to $\left(2M-1\right)$ onto compute node 1, and so on, where M is equal to the ratio of the number of partitions to the number of compute nodes.

The second table in the output displays `Minimum`, `Maximum`, and (where applicable) `Total` values for various partition statistics:

Cell Count
the number of cells in the partitions (corresponding to `Cells` in the per-partition table)

Mean cell count deviation
the deviation of an individual partition cell count from the mean partition cell count

Partition boundary cell count
the number of cells that lie on partition interfaces (corresponding to `I-Cells` in the per-partition table)

Partition boundary cell count ratio
the ratio of the number of cells that lie on partition interfaces to the total number of cells in the partition (corresponding to `Cell Ratio` in the per-partition table)

Face Count
the number of faces in the partitions (corresponding to `Faces` in the per-partition table)

Mean face count deviation
the deviation of an individual partition face count from the mean partition face count

Partition boundary face count
the number of faces that lie on partition interfaces (corresponding to `I-Faces` in the per-partition table)

Partition boundary face count ratio
the ratio of the number of faces that lie on partition interfaces to the total number of faces in the partition (corresponding to `Face Ratio` in the per-partition table)

Partition neighbor count
the number of neighbors for a given partition (corresponding to `Neighbors` in the per-partition table)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Finally, the `Partition Method` and `Stored Partition Count` are displayed.

Your aim is to achieve small magnitudes for `Minimum`, `Maximum`, and `Total` values of the `Partition boundary face count ratio` while maintaining a balanced load (`Mean cell count variation`).

If there is an overset mesh for which the solution has been initialized, an additional partition table with solve and dead cells is included in the partitioning report (see Overset Cell Marks (p. 191) for the definitions of such cells):

```
 >> Overset partition statistics:
     P   Cells Solve-cells Dead-cells  Ext donors
     0      66          53          4          12
     1      66          55          4          10

    ------------------------------------------------------------------------
    Overset Partition Statistics:           Minimum   Maximum   Total
    ------------------------------------------------------------------------
    Cell count                              66        66        132
    Mean cell count deviation               0.0%      0.0%

    Solve cell count                        53        55        108
    Mean solve cell count deviation         -1.9%     1.9%

    Dead cell count                         4         4         8
    Mean dead cell count deviation          0.0%      0.0%

    Ext donors                              10        12        22
    ------------------------------------------------------------------------
    Partition Method                        Metis
    Stored Partition Count                  2
```

34.5.6.2. Examining Partitions Graphically

To further aid interpretation of the partition information, you can draw contours of the mesh partitions (see the figures in Partition Methods (p. 2046)).

Results → Graphics → Contours Edit...

To display the active cell partition or the stored cell partition (which were described above), select **Active Cell Partition** or **Stored Cell Partition** in the **Cell Info...** category of the **Contours Of** drop-down list, and turn off the display of **Node Values** (for details, see Displaying Contours and Profiles (p. 1756) for information about displaying contours).

Important

If you have not already done so in the setup of your problem, you must perform a solution initialization in order to use the **Contours** dialog box.

34.5.7. Load Distribution

If the speeds of the processors that will be used for a parallel calculation differ significantly, you can specify a load distribution for partitioning, using the `load-distribution` text command.

`parallel → partition → set → load-distribution`

For example, if you will be solving on three compute nodes, and one machine is twice as fast as the other two, then you may want to assign twice as many cells to the first machine as to the others (that is, a load vector of (2 1 1)). During subsequent mesh partitioning, partition 0 will end up with twice as many cells as partitions 1 and 2.

For this example, you need to start up ANSYS Fluent such that compute node 0 is the fast machine, since partition 0, with twice as many cells as the others, will be mapped onto compute node 0. Alternatively, in this situation, you could enable the load balancing feature (described in Load Balancing (p. 2044)) to have ANSYS Fluent automatically attempt to discern any difference in load among the compute nodes.

34.5.8. Troubleshooting

When running a calculation using parallel ANSYS Fluent, you may encounter a warning message in the console that reports problems related to the partitioning. The following is an example of such a warning:

```
#AMG# Warning: The global matrix size (1273286) is too large, and may
adversely affect the parallel performance. See the ANSYS Fluent User's
Guide for information on troubleshooting partitioning issues.
```

The following are possible reasons for partitioning problems, along with recommendations for reducing them:

- The presence of solid zones may cause a partition to have a very small amount of fluid cells. For such cases, it is recommended that you partition the mesh with the **Across Zones** option disabled in the **Partitioning and Load Balancing** dialog box, and then click the **Use Stored Partitions** button.
- A partition may have a small number of cells if you have set up a load distribution for partitioning. Such settings should be disabled, by using the `load-distribution` text command (described in Load Distribution (p. 2055)) and entering a value of `1` for each of the previously defined partitions.
- Some model settings (for example, shell conduction) can encapsulate some cells, which may cause difficulties with the coarsening process. To remedy this situation, you can either try a different partitioning method, or you can enable the global coarsening checking criteria with the following rpvar setting:

  ```
  (rpsetvar 'amg/parallel/global-check-coarsening? #t)
  ```

- Coupled walls are encapsulated as part of the shell conduction model and S2S model. If you have partitioning problems, you can try reverting to the encapsulation routine used prior to version 16.0 by disabling the enhanced encapsulation:

  `define` → `models` → `shell-conduction` → `enhanced-encapsulation?`

34.6. Using General Purpose Graphics Processing Units (GPGPUs) With the Algebraic Multigrid (AMG) Solver

You can accelerate the Algebraic Multigrid (AMG) solver inside Fluent using General Purpose Graphics Processing Units (GPGPUs) if suitable hardware is available on your compute machines. When enabled, you can use GPGPU acceleration for AMG computations in a parallel Fluent session on linear systems with up to 5 coupled equations. Using GPGPUs requires HPC licenses. Licensing details can be found in HPC Licensing in the *ANSYS, Inc. Licensing Guide*.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

34.6.1. Requirements

The following GPGPUs are supported:

- For Windows:
  - NVIDIA Tesla K40m
  - NVIDIA Tesla M2075
- For Linux:
  - NVIDIA Tesla K20xm
  - NVIDIA Tesla K40m
  - NVIDIA Tesla K80

The following requirements must be satisfied in order to use this functionality:

- The pressure-based coupled solver must be selected.
- CUDA 6.0 and the required driver must be installed.

In addition, when starting the parallel Fluent session the following conditions must be met:

- The number of solver processes per machine must be equal on all machines.
- The number solver processes per machine must be evenly divisible by the specified number of GPGPUs per machine.

34.6.2. Limitations

GPGPU acceleration is subject to the following limitations:

- When using Fluent in Workbench, the number of GPGPUs cannot be set through:
  - Properties of Setup Cells
  - Properties of Solution Cells
  - Properties of Solution Cells through RSM update
  - Properties of Parameter Sets through RSM update
- GPGPU acceleration will not be used in the following cases:
  - The population balance model is active.
  - The Eulerian multiphase model is active.
  - The system has more than 5 coupled equations.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

34.6.3. Using and Managing GPGPUs

In order to use GPGPUs, you must specify in the Fluent Launcher or with the `-gpgpu=` *ngpgpus* command line option how many GPGPUs are to be used per machine. For details, refer to the following sections:

Starting Parallel ANSYS Fluent Using Fluent Launcher (p. 2012)
Starting Parallel ANSYS Fluent on a Windows System Using Command Line Options (p. 2021)
Starting Parallel ANSYS Fluent on a Linux System (p. 2026)

Once the Fluent session is running, you can view and/or select the available GPGPUs on the system using the following TUI commands:

`parallel/gpgpu/show`
display the available GPGPUs on the system.

`parallel/gpgpu/select`
select the GPGPUs to use. Note that you can only select up to the number of GPGPUs that you specified on the command line or in the Fluent Launcher when starting the session.

By default, GPGPU acceleration is applied automatically to coupled systems and not to scalar systems because scalar systems typically are not as computationally expensive. However, if desired you can enable/disable GPGPU acceleration of the AMG solver for coupled and scalar systems in the following TUI menu:

```
solve/set/amg-options/amg-gpgpu-options/
```

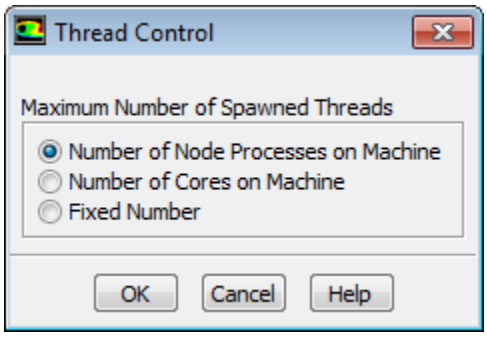
The menu contains commands for each supported equation type allowing you to enable/disable GPGPU acceleration, choose between AMG and FGMRES solvers, and specify various solver options.

34.7. Controlling the Threads

You can control the maximum number of threads on each machine by using the **Thread Control** dialog box (Figure 34.23: The Parallel Connectivity Dialog Box (p. 2059)).

Parallel → General → Thread Control...

Figure 34.22: The Thread Control Dialog Box



You have the following options when using the **Thread Control** dialog box:

- **Number of Node Processes on Machine**

  This is the default option. When this option is chosen, the maximum number of threads on each machine is equal to the number of ANSYS Fluent node processes on each machine.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Number of Cores on Machine**

  When this option is chosen, the maximum number of threads on each machine is equal to the number of cores on the machine. ANSYS Fluent obtains the number of cores from the OS. This may be applicable when the multi-threaded part of the calculation is dominating the computation time, and the continuous phase calculation is relatively small, and you want to take full advantage of the computation resources. For example, if you have a very small case with regard to the number of cells, but a large number of particles to be tracked, you may want to spawn one ANSYS Fluent node process on each machine, but use the maximum number of cores in order to get a good overall performance.

- **Fixed Number**

  When this option is chosen, you may specify the maximum number of threads that can be spawned on each machine in the number-entry box below **Fixed Number**. This may only be applicable when you want to have fine control of the number of threads on each machine; it is not recommended in general.

34.8. Checking Network Connectivity

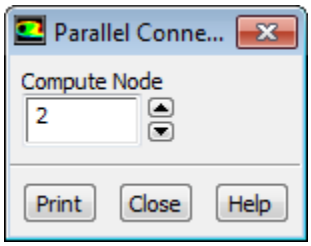
For any compute node, you can print network connectivity information that includes the hostname, architecture, process ID, and ID of the selected compute node and all machines connected to it. The ID of the selected compute node is marked with an asterisk.

The ID for the ANSYS Fluent host process is always `host`. The compute nodes are numbered sequentially starting from `node-0`. All compute nodes are completely connected. In addition, compute node 0 is connected to the host process.

To obtain connectivity information for a compute node, you can use the Parallel Connectivity Dialog Box (p. 2760) (Figure 34.23: The Parallel Connectivity Dialog Box (p. 2059)).

Parallel → Network → Connectivity...

Figure 34.23: The Parallel Connectivity Dialog Box



Indicate the compute node ID for which connectivity information is desired in the **Compute Node** field, and then click the **Print** button. Sample output for compute node 0 is shown below:

```
------------------------------------------------------------------------------------------
ID      Hostname              Core        O.S.        PID     Vendor
------------------------------------------------------------------------------------------
n5      host25                2/64        Linux-64    18909   AMD Opteron(tm) 6278
n2      host25                1/64        Linux-64    18908   AMD Opteron(tm) 6278
n4      host24                2/64        Linux-64    32939   AMD Opteron(tm) 6278
n1      host24                1/64        Linux-64    32938   AMD Opteron(tm) 6278
host    host23                -           Linux-64    38427   AMD Opteron(tm) 6278
n3      host23                2/64        Linux-64    38755   AMD Opteron(tm) 6278
n0*     host23                1/64        Linux-64    38754   AMD Opteron(tm) 6278
```

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

34.9. Checking and Improving Parallel Performance

Fluent offers several tools to help you optimize the performance of your parallel computations. You can check the utilization of your hardware using the parallel check feature. To determine how well the parallel solver is working, you can measure computation and communication times, and the overall parallel efficiency, using the performance meter. You can also control the amount of communication between compute nodes in order to optimize the parallel solver, and take advantage of the automatic load balancing feature of ANSYS Fluent.

Information about checking and improving parallel performance is provided in the following sections:

34.9.1. Parallel Check
34.9.2. Checking Parallel Performance
34.9.3. Optimizing the Parallel Solver
34.9.4. Clearing the Linux File Cache Buffers

34.9.1. Parallel Check

You can use the **Check** command in the **Parallel** ribbon tab to check various factors that affect parallel performance. Checks are performed to identify the following issues:

- CPU cores are overloaded
- CPU clock is throttled
- System memory usage is too high
- A faster interconnect is available
- Partitions are imbalanced (if a valid mesh is loaded)

34.9.2. Checking Parallel Performance

The performance meter allows you to report the wall clock time elapsed during a computation, as well as message-passing statistics. Since the performance meter is always enabled, you can access the statistics by displaying them after the computation is completed. To view the current statistics, click **Usage** in the **Parallel** ribbon tab (**Timer** group box).

Parallel → **Timer** → **Usage**

Performance statistics will be displayed in the console.

To clear the performance meter so that you can eliminate past statistics from the future report, click **Reset** in the **Parallel** ribbon tab (**Timer** group box).

Parallel → **Timer** → **Reset**

The following example demonstrates how the current parallel statistics are displayed in the console:

```
Performance Timer for 1 iterations on 4 compute nodes
Average wall-clock time per iteration:           4.901 sec
Global reductions per iteration:                   408 ops
Global reductions time per iteration:            0.000 sec (0.0%)
Message count per iteration:                       801 messages
Data transfer per iteration:                     9.585 MB
LE solves per iteration:                           12 solves
LE wall-clock time per iteration:                2.445 sec (49.9%)
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
  LE global solves per iteration:                    27 solves
  LE global wall-clock time per iteration:        0.246 sec (5.0%)
  AMG cycles per iteration:                          64 cycles
  Relaxation sweeps per iteration:                 4160 sweeps
  Relaxation exchanges per iteration:               920 exchanges

  Total wall-clock time:                          4.901 sec
```

A description of the parallel statistics is as follows:

- `Average wall-clock time per iteration` describes the average real (wall clock) time per iteration.
- `Global reductions per iteration` describes the number of global reduction operations (such as variable summations over all processes). This requires communication among all processes.

  A global reduction is a collective operation over all processes for the given job that reduces a vector quantity (the length given by the number of processes or nodes) to a scalar quantity (for example, taking the sum or maximum of a particular quantity). The number of global reductions cannot be calculated from any other readily known quantities. The number is generally dependent on the algorithm being used and the problem being solved.
- `Global reductions time per iteration` describes the time per iteration for the global reduction operations.
- `Message count per iteration` describes the number of messages sent between all processes per iteration. This is important with regard to communication latency, especially on high-latency interconnects.

  A message is defined as a single point-to-point, send-and-receive operation between any two processes. This excludes global, collective operations such as global reductions. In terms of domain decomposition, a message is passed from the process governing one subdomain to a process governing another (usually adjacent) subdomain.

  The message count per iteration is usually dependent on the algorithm being used and the problem being solved. The message count and the number of messages that are reported are totals for all processors.

  The message count provides some insight into the impact of communication latency on parallel performance. A higher message count indicates that the parallel performance may be more adversely affected if a high-latency interconnect is being used. Ethernet has a higher latency than Myrinet or Infiniband. Therefore, a high message count will more adversely affect performance with Ethernet than with Infiniband.

  To check the latency of the overall cluster interconnect, refer to Checking Latency and Bandwidth (p. 2062).
- `Data transfer per iteration` describes the amount of data communicated between processors per iteration. This is important with respect to interconnect bandwidth.

  Data transfer per iteration is usually dependent on the algorithm being used and the problem being solved. This number generally increases with increases in problem size, number of partitions, and physics complexity.

  The data transfer per iteration may provide some insight into the impact of communication bandwidth (speed) on parallel performance. The precise impact is often difficult to quantify because it is dependent on many things including: ratio of data transfer to calculations, and ratio of communication bandwidth to CPU speed. The unit of data transfer is a byte.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To check the bandwidth of the overall cluster interconnect, refer to Checking Latency and Bandwidth (p. 2062).

- `LE solves per iteration` describes the number of linear systems being solved per iteration. This number is dependent on the physics (non-reacting versus reacting flow) and the algorithms (pressure-based versus density-based solver), but is independent of mesh size. For the pressure-based solver, this is usually the number of transport equations being solved (mass, momentum, energy, and so on).
- `LE wall-clock time per iteration` describes the time (wall-clock) spent doing linear equation solvers (that is, multigrid).
- `LE global solves per iteration` describes the number of solutions on the coarsest level of the AMG solver where the entire linear system has been pushed to a single processor (`n0`). The system is pushed to a single processor to reduce the computation time during the solution on that level. Scaling generally is not adversely affected because the number of unknowns is small on the coarser levels.
- `LE global wall-clock time per iteration` describes the time (wall-clock) per iteration for the linear equation global solutions.
- `AMG cycles per iteration` describes the average number of multigrid cycles (V, W, flexible, and so on) per iteration.
- `Relaxation sweeps per iteration` describes the number of relaxation sweeps (or iterative solutions) on all levels for all equations per iteration. A relaxation sweep is usually one iteration of Gauss-Siedel or ILU.
- `Relaxation exchanges per iteration` describes the number of solution communications between processors during the relaxation process in AMG. This number may be less than the number of sweeps because of shifting the linear system on coarser levels to a single node/process.
- `Time-step updates per iteration` describes the number of sub-iterations on the time step per iteration.
- `Time-step wall-clock time per iteration` describes the time per sub-iteration.
- `Total wall-clock time` describes the total wall-clock time.

The most relevant quantity is the `Total wall clock time`. This quantity can be used to gauge the parallel performance (speedup and efficiency) by comparing this quantity to that from the serial analysis (the command line should contain `-t1` in order to obtain the statistics from a serial analysis).

34.9.2.1. Checking Latency and Bandwidth

You can check the latency and bandwidth of the overall cluster interconnect, to help identify any issues affecting ANSYS Fluent scalability, by clicking **Latency** and **Bandwidth** in the **Parallel** ribbon tab (**Network** group box).

Parallel → Network → Latency

Depending on the number of machines and processors being used, a table containing information about the communication speed for each node will be displayed in the console. The table will also summarize the minimum and maximum latency between two nodes.

Consider the following example when checking for latency:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Latency (usec) with 1000 samples [1.83128 sec]
 ----------------------------------------
 ID       n0     n1     n2    n3    n4    n5
 ----------------------------------------
 n0            48.0   48.2  48.2  48.3   *50
 n1    48.0           48.2  48.3  48.3   *48
 n2    48.2    48.2         48.8  49.1   *53
 n3    48.2    48.3    *49        48.6  48.5
 n4    48.3    48.3   49.1  48.6         *50
 n5    49.7    48.5    *53  48.5  49.7

 ----------------------------------------
 Min: 47.9956 [n0<-->n1]
 Max: 52.6836 [n5<-->n2]
 ----------------------------------------
```

Important

In the above table, (*) is the maximum value in that row. The smaller the latency, the better.

Six processors (n0 to n5) are spawned. The latency between n0 and n1 is 48.0 μs. Similarly, the latency between n1 and n2 is 48.2 μs. The minimum latency occurs between n0 and n1 and the maximum latency occurs between n2 and n5, as noted in the table. Checking the latency is particularly useful when you are not seeing expected speedup on a cluster.

Parallel → Network → Bandwidth

In addition to checking for latency, you can check your bandwidth. A table containing information about the amount of data communicated within one second between two nodes is displayed in the console. The table will also summarize the minimum and maximum bandwidth between two nodes.

Consider the following example when checking for bandwidth:

```
Bandwidth (MB/s) with 5 messages of size 4MB [4.36388 sec]
 ----------------------------------------
 ID      n0     n1     n2     n3     n4     n5
 ----------------------------------------
 n0          111.8    *55  111.8   97.5  101.3
 n1   111.8          69.2   98.7  111.7    *51
 n2    54.7   69.2          72.9  104.8    *45
 n3   111.8   98.7   72.9          64.0    *45
 n4    97.6  111.7  104.8    *64          76.9
 n5   101.2   50.9   45.5    *45    76.9

 ----------------------------------------
 Min: 45.1039 [n5<-->n3]
 Max: 111.847 [n0<-->n3]
 ----------------------------------------
```

Important

In the above table, (*) is the minimum value in that row. The larger the bandwidth, the better.

The bandwidth between n0 and n1 is 111.8 MB/s. Similarly, the bandwidth between n1 and n2 is 69.2 MB/s. The minimum amount of bandwidth occurs between n3 and n5 and the maximum occurs between n0 and n3, as noted in the table. Checking the bandwidth is particularly useful when you cannot see good scalability with relatively large cases.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

34.9.3. Optimizing the Parallel Solver

34.9.3.1. Increasing the Report Interval

In ANSYS Fluent, you can reduce communication and improve parallel performance by increasing the report interval for residual printing/plotting or other solution monitoring reports. You can modify the value for **Reporting Interval** in the Run Calculation Task Page (p. 2537).

Solution → Run Calculation Calculate...

Important

Note that you will be unable to interrupt iterations until the end of each report interval.

34.9.3.2. Accelerating View Factor Calculations for General Purpose Computing on Graphics Processing Units (GPGPUs)

View factor computations can be accelerated through the `viewfac_acc` utility that uses a combination of MPI/OpenMP/OpenCL models to speed up view factor computations. Irrespective of the number of MPI processes launched, only one MPI process/machine is used for computing view factors. On each machine, one MPI process spawns several OpenMP threads that actually compute the view factors. Since only one MPI process is required per machine, it is recommended that you start just one MPI process per machine and specify the number of OpenMP threads to use when running the utility outside of ANSYS Fluent. With fewer MPI processes, the system memory usage is reduced as well. When running the utility from inside ANSYS Fluent, the number of `viewfac_acc` processes will be same as the number of ANSYS Fluent processes. If OpenCL-capable GPUs are available, then a portion of the view factor computations are done on GPUs using OpenCL, to further speed up the computation. At present, this capability is limited to the hemicube method with the cluster-to-cluster option on lnamd64 and win64 machines.

When using the utility `viewfac_acc` outside of ANSYS Fluent, you can specify the following command line options:

- `-cpu` # (default = -2) The number of OpenMP threads to launch per machine.
  - -2: The number of MPI processes.
  - -1: The total number of logical CPU cores.
  - 0: No CPU used.
  - n: Up to maximum n.
- `-gpu` # (default = 1) The number of GPU devices to use per machine.
  - –1: The total number of GPU devices.
  - 0: No GPU used.
  - n: Up to maximum n.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `-gpu_cpu_ratio` # (default = 2.0) The ratio of the work load on 1 GPU vs 1 CPU OpenMP thread. This is based on the time consumed by the GPU and the CPU. At the end of the view factor computations, a recommendation is printed for the GPU/CPU work load ratio to use in future computations.

When using the `viewfac_acc` utility from inside an ANSYS Fluent session, use the `/define/models/radiation/s2s-parameters/compute-clusters-and-vf-accelerated` text interface (TUI) command. You will only be prompted for the `-gpu` and `-gpu_cpu_ratio` options (as described previously) and not the `-cpu` option, as the number of OpenMP threads to launch per machine can be specified in the Thread Control Dialog Box (p. 2811) (see Controlling the Threads (p. 2058) for details).

In order to use the GPU for view factor computations, the OpenCL library should be accessible through the appropriate environment variable (`LD_LIBRARY_PATH` on lnamd64 or `%path%` on win64). By default on lnamd64, `/usr/lib64` is searched, but if the library is installed in another location, then that location should be specified in the `LD_LIBRARY_PATH` variable.

View factor computations can also be accelerated through the `raytracing_acc` utility that uses the NVIDIA Optix library for tracing the rays. The GPU available on the machine running the host process is used in such a scenario, except in a mixed Windows-Linux simulation where the GPU on node-0 is used. An NVIDIA GPU along with CUDA 6.0 is required for using `raytracing_acc`. At present, this utility is available only on lnamd64 (Red Hat Enterprise Linux 5/6, and SUSE Linux Enterprise Server 11) and win64 (Windows 7) machines for 3D problems. In order to use the utility, the CUDA 6.0 library should be accessible through the appropriate environment variable (`LD_LIBRARY_PATH` on lnamd64 or `%path%` on win64).

When using the `raytracing_acc` utility from outside an ANSYS Fluent session, the command line is `utility raytracing_acc <input_cluster_file> [output_s2s_file(optional)]`.

When using the `raytracing_acc` utility from inside an ANSYS Fluent session, use the `/define/models/radiation/s2s-parameters/compute-clusters-and-vf-accelerated` text user interface (TUI) command.

34.9.3.3. Accelerating Discrete Ordinates (DO) Radiation Calculations

An accelerated discrete ordinate (DO) radiation solver is available on Linux when running a parallel Fluent session. While it is computationally faster than the standard DO solver, it may take a larger number of iterations to converge.

After you have selected the DO model in the **Radiation Model** dialog box, you can enable the accelerated DO solver by using the following text command:

`define` → `models` → `radiation` → `do-acceleration?`

If NVIDIA GPGPUs are enabled in the Fluent session, this solver will accelerate the DO computations by using the GPGPUs. In the absence of GPGPUs, this solver can still be used with the CPU cores to accelerate the DO computations. Note that the accelerated DO solver uses the first-order upwind scheme and an explicit relaxation of 1.0.

The accelerated DO solver is incompatible with some models and settings; when necessary, Fluent will automatically revert to the standard DO solver when the calculation is started and print a message about the conflict.

If you plan to use GPGPUs with the accelerated DO solver, it is recommended that you run NVIDIA's multi-process server (MPS) before launching ANSYS Fluent using the following command: `nvidia-`

`cuda-mps-control -d`. It is known to improve the robustness and performance of the GPGPU computations with the multiple Fluent processes.

34.9.4. Clearing the Linux File Cache Buffers

Processing performance can significantly decrease when the file cache buffers of a Linux machine are full. While this is true in serial, it is more often a concern when solving large cases in parallel, particularly when using AMD processors. If you see a performance decrease even though the case / machine setup has not changed, that is an indication that the file cache buffers may be to blame.

The filling of the file cache buffers can happen over a period of time as a result of input-output activity. Even after the ANSYS Fluent session is exited, by default the operating system does not free up the file cache buffers immediately (unless the operating system is unable to satisfy a malloc subroutine request with available free memory). During memory allocation for a parallel case, this can result in the allocation of memory from a different NUMA domain, and consequently can have a significant impact on performance.

To resolve this issue on Linux machines, you must first ensure that all of the relevant machines are idle (so that you do not adversely affect any jobs that are running). Then you can clear the file cache buffers by performing one of the following actions:

- Include the `-cflush` option when launching ANSYS Fluent from the command line.

  This option ensures that the file cache buffers are flushed in a separate operation. While this process may take a few minutes to complete (depending on the total memory of the system), it does not require you to have root privileges.

or

- Enter the `(drop-cache)` Scheme command (either in the ANSYS Fluent console or through your journal file) after launching Fluent but before you read the case file.

  This command will instantaneously clear the pagecaches, dentries, and inodes.

  Important

  Note that in order to use the `(drop-cache)` command, you must have sudo administrative privileges for the `/sbin/sysctl vm.drop_caches=3` command.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 35: Design Analysis and Optimization

In this chapter, the setup and use of the adjoint solver and the mesh morpher/optimizer in ANSYS Fluent are described. Both tools offer shape optimization capabilities, but the scope is complementary. The fundamental difference in the technology is that the adjoint solver uses gradient-based optimization by calculating derivatives, whereas the mesh morpher/optimizer uses a strategy that is not based on gradients to decide design updates.

The tools have a common approach to change a design by morphing the computational mesh. This has the benefit that you need not return to the original geometry in order to update the design for the purposes of the optimization.

The two key considerations when deciding whether to use the adjoint solver or the mesh morpher/optimizer are as follows:

- Is the flow physics / observable of interest supported by the adjoint solver? (For details about adjoint capabilities, see the following sections: Basic Assumptions and Consistency Checks (p. 2082) and General Observables (p. 2069)) If not, then the mesh morpher/optimizer is the better option, since there are very few limitations in the supported physics and choice of quantity of interest with this approach. It is important to verify that the quantity of interest can be specified as an adjoint solver observable.

- Is the dimension of the design space high? Detailed shape optimization of 3D geometries can involve design spaces with large dimensions (thousands of degrees of freedom or more). The adjoint solver is very effective in this scenario, whereas the use of the mesh morpher/optimizer becomes computationally prohibitive when the number of degrees of freedom exceeds more than, say, 10–20.

Details are provided in the following sections:

35.1. The Adjoint Solver
35.2. Using the Adjoint Solver
35.3. The Mesh Morpher/Optimizer
35.4. Using the Mesh Morpher/Optimizer

Note that the Fluent Tutorial Guide presents a tutorial that uses the adjoint solver. (To access tutorials and their input files on the ANSYS Customer Portal, go to http://support.ansys.com/training.)

35.1. The Adjoint Solver

An adjoint solver is a specialized tool that extends the scope of the analysis provided by a conventional flow solver by providing detailed sensitivity data for the performance of a fluid system.

In order to perform a simulation using the ANSYS Fluent standard flow solvers, a user supplies system geometry in the form of a computational mesh, specifies material properties and physics models, and configures boundary conditions of various types. The conventional flow solver, once converged, provides a detailed data set that describes the flow state governed by the flow physics that are being modeled. Various postprocessing steps can be taken to assess the performance of the system.

If a change is made to any of the data that defines the problem, then the results of the calculation can change. The degree to which the solution changes depends on how sensitive the flow is to the partic-

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ular parameter that is being adjusted. Indeed, the derivative of the solution data with respect to that parameter quantifies this sensitivity to first order. Determining these derivatives is the domain of *sensitivity analysis*.

There is a large collection of derivative data that can be computed for a fluid system, given the extensive set of input data that is required, and the extensive flow data that is produced. The matrix of derivatives of output data with respect to input data can be vast. Depending upon the goal of the analysis only a portion of this derivative data may be needed for engineering analysis and decision-making.

The adjoint solver accomplishes the remarkable feat of calculating the derivative of a single engineering observation with respect to a very large number of input parameters simultaneously via a *single computation*. The engineering observation could be a measure of the system performance, such as the lift or drag on an airfoil, or the total pressure drop through a system. Most importantly, the derivatives with respect to the geometric shape of the system are found.

Understanding such sensitivities in a fluid system can provide extremely valuable engineering insight. A system that is highly sensitive may exhibit strong variability in performance due to small variations in manufacturing or variations in the environment in which it is operating. Alternatively, high-sensitivity may be leveraged for fluid control, with a small actuator being able to induce strong variations in behavior. Yet another perspective is that sensitivity of a performance measure implies that the device in question is not fully optimized and there is still room for improvement—assuming that constraints do not preclude further gains.

The sensitivities of a fluid system provided by an adjoint solver satisfy a central need in gradient-based shape optimization. This makes an adjoint solver a unique and powerful engineering tool for design optimization.

Adjoint data can also play a role in improving solver numerics. Regions of high sensitivity are indicative of areas in the flow where discretization errors can potentially have a strong effect. This information can be used to guide how best to refine a mesh to improve flow solution accuracy.

The process of computing an adjoint solution resembles that for a standard flow calculation in many respects. The adjoint solver solution advancement method is specified, residual monitors configured, and the solver is initialized and run through a sequence of iterations to convergence. One notable difference is that a scalar-valued observation is selected as being of interest prior to starting the adjoint calculation.

Once the adjoint solution is converged the derivative of the observable with respect to the position of each and every point on the surface of the geometry is available, and the sensitivity of the observation to specific boundary condition settings can be found. This remarkable feature of adjoint solutions has been known for hundreds of years, but only in the last 25 years has the significance for computational physics analysis been recognized widely.

The power of this methodology is highlighted when an alternative for assembling the same information is considered. Imagine a sequence of flow calculations in which each point on an airfoil surface is moved in turn a set small distance in the surface-normal direction, and the flow and drag recomputed. If there are N points on the surface, then N flow calculations are required to build the data set. Considering that the same data is provided by a *single* adjoint computation, the adjoint approach has an enormous advantage. Remember that for even modest 3D flow computations there may be many thousands of coordinates or more on a surface.

Once the adjoint is computed it can be used to guide intelligent design modifications to a system. After all, the adjoint sensitivity data provides a map across the entire surface of the geometry of the effect of moving the surface. Design modifications can be most effective if made in regions of high sensitivity

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

since small changes will have a large effect upon the engineering quantity of interest. This principle of making changes to a system in proportion to the local sensitivity is the foundation for the simple gradient algorithm for design optimization.

Once a candidate change in shape or other boundary condition has been selected, the effect of that change can be estimated using the computed derivative data. This amounts to a first order extrapolation using a Taylor series expansion around the baseline flow state. Clearly if a modification is chosen that is large enough that nonlinear effects become important then the accuracy of the predicted change cannot be guaranteed.

35.1.1. General Observables

Several observables are available and serve as the foundation for specifying the quantity that is of interest for the computation. Basic quantities such as forces and moments can be defined and each is given a name. The following types of observables are available:

- Force: the aerodynamic force in a specified direction on one or more walls.
- Moment of force: the aerodynamic moment about a specified moment center and moment axis. The integration is made over a specified collection of wall zones. Note that in 2D, the moment axis is normal to the plane of the flow.
- Swirl: the moment of the mass flow (with velocity $\underline{u}$) relative to an axis defined by a point, $\underline{r_c}$, and a direction $\underline{d}$. The two options are:
  - Swirl integral

$$\int_V \rho(\underline{r}\times\underline{u})\cdot\underline{d}dV \tag{35.1}$$

    where V denotes the volume over which the integration is made, and $\underline{r}$ denotes the relative position to the point $\underline{r_c}$.

  - Normalized swirl integral

$$\frac{\int_V \rho(\underline{r}\times\underline{u})\cdot\underline{d}dV}{\int_V \rho(\underline{r}\times(\underline{d}\times\underline{r}))\cdot\underline{d}dV} \tag{35.2}$$

    Note

    These integrals can be used to construct quantities such as tumble ratio that are of significance for internal combustion engine analysis.

- pressure drop between an inlet (or group of inlets) and an outlet (or group of outlets)

  The total pressure is computed by area-weighted averaging of the total pressure values on selected surfaces.
- fixed value: a simple fixed value can be specified and used in the assembly of the observables. This is convenient when some scaling or normalization of the observable is desired. It must be noted that this value is treated strictly as a constant for the purposes of any derivative calculations.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- surface integral: a variety of surface integrals can be constructed for a specified field variable on a set of user-selected surfaces:

  – Facet sum: a simple sum of the field value on each facet of the computational mesh

  $$\sum_f \varphi_f \tag{35.3}$$

  – Facet average: the facet sum is divided by the total number of facets, N_f

  $$\overline{\varphi} = \frac{\sum_f \varphi_f}{N_f} \tag{35.4}$$

  – Facet variance: the sum of the squares of the deviations from the facet average

  $$\frac{\sum_f (\varphi_f - \overline{\varphi})^2}{N_f} \tag{35.5}$$

  – Integral: the sum of the field value on each face multiplied by the face area

  $$\sum_f \left|\underline{A}_f\right| \varphi_f \tag{35.6}$$

  – Area-weighted average: the integral divided by the total face area

  $$\frac{\sum_f \left|\underline{A}_f\right| \varphi_f}{\sum_f \left|\underline{A}_f\right|} \tag{35.7}$$

  – Area-weighted variance: the sum of the squares of the deviations from the area-weighted average, divided by the total face area

  $$\frac{\sum_f \left|\underline{A}_f\right| (\varphi_f - \overline{\varphi})^2}{\sum_f \left|\underline{A}_f\right|} \tag{35.8}$$

  – Mass-weighted integral: the sum of the field value on each face, weighted by the magnitude of the local mass flow through the face.

  $$\sum_f \left|\rho \underline{u}_f \cdot \underline{A}_f\right| \varphi_f \tag{35.9}$$

  – Mass-weighted average integral: the mass-weighted integral divided by the sum of the magnitudes of the local mass flow rates through the faces on which the integral is defined.

  $$\frac{\sum_f \left|\rho \underline{u}_f \cdot \underline{A}_f\right| \varphi_f}{\sum_f \left|\rho \underline{u}_f \cdot \underline{A}_f\right|} \tag{35.10}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– Mass-weighted variance: the mass-weighted integral of the square of the deviation from the mass-weighted average, divided by the sum of the magnitudes of the local mass flow rates through the faces on which the integral is defined.

$$\frac{\sum_f |\rho \underline{u}_f \cdot \underline{A}_f| \left(\varphi_f - \overline{\varphi}\right)^2}{\sum_f |\rho \underline{u}_f \cdot \underline{A}_f|} \tag{35.11}$$

– Flow-rate weighted: the rate at which the field is convected through the defined surfaces.

$$\sum_f \left(\rho \underline{u}_f \cdot \underline{A}_f\right) \varphi_f \tag{35.12}$$

– The field variables that can be used are:

→ Pressure

→ Total pressure

→ Mass flow per unit area

→ Temperature (when adjoint energy is enabled)

→ Heat flux (when adjoint energy is enabled)

• volume integral: you can compute a variety of volume integrals of a chosen field variable.

– Volume: computes the total volume of the selected zones as a summation of the individual cell volumes.

$$\int dV = \sum_{i=1}^{n} |V_i| \tag{35.13}$$

– Sum: computes the sum of a chosen field variable over the selected zones.

$$\sum_{i=1}^{n} |\varphi_i| \tag{35.14}$$

– Volume Integral: is calculated by summing the product of cell volume and the selected field variable over the selected zones.

$$\int \varphi dV = \sum_{i=1}^{n} \varphi_i |V_i| \tag{35.15}$$

– Volume-Weighted Average: is computed by dividing the volume integral of a selected field variable (Equation 35.15 (p. 2071)) by the total volume of the selected zones.

$$\int \varphi dV = \frac{1}{V} \sum_{i=1}^{n} \varphi_i |V_i| \tag{35.16}$$

– Volume Variance: is the volume-weighted integral of the square of the deviation from the volume-weighted average (Equation 35.16 (p. 2071)), divided by the total volume of the selected zone.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\frac{\sum_{i=1}^{n}|V_i|\left(\varphi_i-\overline{\varphi}\right)^2}{\sum_{i=1}^{n}|V_i|} \tag{35.17}$$

– Mass Integral: is computed by summing the product of density, cell volume, and the chosen field variable.

$$\int \rho\varphi dV=\sum_{i=1}^{n}\rho_i\varphi_i|V_i| \tag{35.18}$$

– Mass: is computed by summing the product of cell density and cell volume.

$$\int \rho dV=\sum_{i=1}^{n}\rho_i|V_i| \tag{35.19}$$

– Mass-Weighted Average: is computed by dividing the Mass Integral of the chosen field variable (Equation 35.18 (p. 2072)) by the total Mass (Equation 35.19 (p. 2072))

$$\frac{\int \rho\varphi dV=\sum_{i=1}^{n}\rho_i\varphi_i|V_i|}{\int \rho dV=\sum_{i=1}^{n}\rho_i|V_i|} \tag{35.20}$$

– Mass Variance: is computed as the mass-weighted summation of the square of the deviation of the chosen field variable from its mass-weighted average (Equation 35.20 (p. 2072)) divided by the total mass (Equation 35.19 (p. 2072)).

$$\frac{\sum_{i=1}^{n}\rho_i|V_i|\left(\varphi_i-\overline{\varphi}\right)^2}{\sum_{i=1}^{n}\rho_i|V_i|} \tag{35.21}$$

The volume integrals listed above can be computed for the following field variables:

– Pressure

– Total Pressure

– Velocity

– Velocity Magnitude

– Vorticity

– Vorticity Magnitude

– Turbulence Production: $\frac{\partial u_i}{\partial x_j}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)$

35.1.2. General Operations

Several operations are available to combine observables in various ways, if needed, to make a wide variety of compound observables. The following operations are available:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- ratio of two quantities
- product of two quantities
- linear combination of N quantities $q_0 \ldots q_{N-1}$, with constant coefficients $a_0 \ldots a_{N-1}$ raised to various powers $v_0 \ldots v_{N-1}$, plus a constant offset c.

$$c + \sum_{i=0}^{N-1} a_i q_i^{v_i} \tag{35.22}$$

- arithmetic average of N quantities $q_0 \ldots q_{N-1}$

$$\overline{q} = \frac{\sum_{i=0}^{N-1} q_i}{N} \tag{35.23}$$

- mean variance of N quantities

$$\frac{\sum_{i=0}^{N-1} (q_i - \overline{q})^2}{N} \tag{35.24}$$

- unary operation (sin, cosine, and so on) on observable value q:
  - $1/q$
  - $|q|$
  - $\sqrt{q}$
  - $\sin q$
  - $\cos q$
  - $\tan q$
  - $\sin^{-1} q$
  - $\cos^{-1} q$
  - $\tan^{-1} q$
  - $\ln q$
  - $\log 10\ q$

35.1.3. Discrete Versus Continuous Adjoint Solver

When an adjoint solver is developed there is a key design decision that is made regarding the implementation, namely whether to use a *continuous* or a *discrete* approach. Both approaches are intended to compute the same sensitivity data—however, the approaches are remarkably different.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

While a user of the adjoint solver may experience no particular difference in workflow for the two separate solver types, it is important to be aware that there are two primary classes of adjoint solver and the approach chosen can have implications for the accuracy of the results.

A *continuous adjoint solver* relies heavily on mathematical properties of the partial-differential equations that define the physics of the problem. In this case those equations are the Navier-Stokes equations. With this approach an adjoint partial differential equation set is formulated explicitly and is accompanied by adjoint boundary conditions that are also derived mathematically. Only after this derivation is complete can the adjoint partial differential equations be discretized and solved, often with extensive re-use of existing solver machinery. This class of solver was implemented by ANSYS Fluent as a research effort.

Such a solver has the benefit that it is decoupled largely from the original flow solver. They share only the fact that they are based on the Navier-Stokes equations. The process of discretizing and solving the partial differential equations in each case could in principle be very different indeed. While this flexibility may be appealing it can also be the downfall of the approach. Inconsistencies in modeling, discretization and solution approaches can pollute the sensitivity information significantly, especially for problems with wall functions and complex engineering configurations such as those of interest to ANSYS Fluent users.

The continuous adjoint approach can be effective for some classes of problems. However, until there is a significant advance in handling some key challenges, ANSYS Inc. has concluded that it is an unsuitable approach for meeting the needs of our broad client base for the classes of problems of interest to them.

A *discrete adjoint solver* is based not on the form of the partial differential equations governing the flow, but the particular discretized form of the equations used in the flow solver itself. The sensitivity of the discretized equations forms the basis for the sensitivity calculation. In this approach the adjoint solver is much more tightly tied to the specific implementation of the original flow solver. This has been observed to yield sensitivity data that provides valuable engineering guidance for the classes of problem of interest to ANSYS Fluent users, including problems with wall functions.

For the above reasons, the discrete adjoint approach has been adopted for the ANSYS Fluent adjoint solver.

35.1.4. Discrete Adjoint Solver Overview

As discussed in the previous section, a discrete adjoint approach to solving the adjoint problem has been adopted, since it is believed to provide the most useful engineering sensitivity data for the classes of problems of interest to ANSYS clients.

An adjoint method can be used to compute the derivative of an observation of interest for the fluid system with respect to all the user-specified parameters, *with any changes that arise in the flow variables themselves eliminated*. There are three key ingredients to consider when developing the method:

1. All of the user-specified inputs:

   - All values set by a user in the boundary condition panels for each boundary in the problem.
   - The computational mesh. More specifically the locations of the nodes of the mesh and how they define the edges, faces, and ultimately the cells used in the finite-volume computation. This includes both interior as well as boundary nodes.
   - Material properties.
   - Model parameters such as model coefficients for turbulence models.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that the settings that define the problem are being distinguished from settings that define how the solution advancement is to be performed to converge the problem. Only the former are of interest here. For the sake of clarity, let us denote the vector of all of the values in the list by $\underline{c}$. These are considered to be the control variables for the problem, that is, the variables that a user can set explicitly that affect the solution.

It is worth noting that the topological definition of the mesh is fixed—it is not considered to be a control variable here. The effect of collapsing cells or remeshing on the flow solution is not addressable without further development in view of the discrete changes that are implied. This topic is beyond the scope of the current document.

2. The governing equations for the fluid system:

   The main effort in a flow computation is in the determination of the flow state, namely the velocity, pressure, density and possibly other fluid-related variables. For a cell-centered finite-volume scheme, the flow state is defined at the cell centroids by a vector of real values. In the simplest case these values are the pressure and flow velocity components. Let the vector of the variables in the ν th cell be denoted here by $\underline{q}^{\nu}$.

   At convergence the flow variables satisfy

$$\mathcal{R}_i^{\mu}\left(\underline{q}^0,\underline{q}^1,\ldots,\underline{q}^{M-1};\underline{c}\right)=0,\ \mu=0,\ldots,M-1,\ i=0,\ldots,L-1 \tag{35.25}$$

   where M is the number of cells in the problem, and there are L conditions on each cell. This expression is a compact way of denoting conservation of mass and momentum and other constraints.

3. The engineering observation of interest:

   Let

$$\mathcal{J}\left(\underline{q}^0,\underline{q}^1,\ldots,\underline{q}^{M-1};\underline{c}\right) \tag{35.26}$$

   denote a scalar of interest that depends both on the flow state and perhaps directly on the control variables. It is assumed that the observable is differentiable with respect to both the flow and the controls. The inclusion of the control variables here is essential since in many cases the mesh geometry is included directly in the evaluation of the observable. For example, the evaluation of the force on a boundary involves the wall normal and face areas, which change when mesh nodes are moved.

The goal is to determine the sensitivity of the observation with respect to the user-specified control variables. What makes defining this relationship more challenging is the fact that changing the user inputs changes the flow, which indirectly changes the engineering observation. The adjoint method has a specific role in managing this chain of influences by providing a mechanism for eliminating the specific changes that happen in the flow whenever the inputs change.

If a variation δc_j is introduced into the control variables then a linearization of the governing equations (Equation 35.25 (p. 2075)) shows that the variations in the flow state δq_j^{ν} must satisfy

$$\frac{\partial R_i^{\mu}}{\partial q_j^{\nu}}\delta q_j^{\nu}=-\frac{\partial R_i^{\mu}}{\partial c_j}\Big|_q \delta c_j,\ \mu=0,\ldots,M-1,\quad i=0,\ldots,L-1. \tag{35.27}$$

where there is an implied summation over j and ν, and $\Big|_q$ denotes that the flow solution is held constant while the derivative is taken.

Meanwhile, if both the control variables and the flow state change, then the observation will change:

$$\delta \mathcal{J} = \frac{\partial \mathcal{J}}{\partial q_j^\nu} \delta q_j^\nu + \left. \frac{\partial \mathcal{J}}{\partial c_j} \right|_q \delta c_j. \tag{35.28}$$

The particular way in which the flow responds to the changes in the control variables can be computed using (Equation 35.27 (p. 2075)) only after specific changes, δc_j, have been chosen. It is prohibitive to consider solving (Equation 35.27 (p. 2075)) for more than a handful of prescribed changes δc_j because of the excessive computing time that would be needed. However, when redesigning the shape of parts of a system there may be pressure to explore a large number of candidate modifications. This conflict is reconciled by eliminating the variations of the flow solution from the expression (Equation 35.28 (p. 2076)) and producing an explicit relationship between changes in the control variables and the observation of interest.

This is accomplished by taking a weighted linear combination of the linearized governing equations (Equation 35.27 (p. 2075)) in a very particular way. A set of adjoint variables $\tilde{q}^\mu$ is introduced with a one-to-one correspondence with the governing equations (Equation 35.25 (p. 2075)). This results in a relationship

$$\left[\tilde{q}_i^\mu \frac{\partial R_i^\mu}{\partial q_j^\nu} \right] \delta q_j^\nu = - \tilde{q}_i^\mu \left. \frac{\partial R_i^\mu}{\partial c_j} \right|_q \delta c_j. \tag{35.29}$$

The term in square brackets on the left is now matched to the coefficient for the variation in the flow in (Equation 35.28 (p. 2076)) in order to define values for the adjoint variables:

$$\frac{\partial R_i^\mu}{\partial q_j^\nu} \tilde{q}_i^\mu = \frac{\partial \mathcal{J}}{\partial q_j^\nu}. \tag{35.30}$$

These are the discrete adjoint equations, and the solution of this system that is the primary goal for the adjoint solver. It is important to recognize that these equations have not been derived. They are defined in this way with a specific goal in mind—the elimination from (Equation 35.28 (p. 2076)) of the perturbations to the flow field as shown below:

$$\begin{aligned} \delta \mathcal{J} &= \frac{\partial \mathcal{J}}{\partial q_j^\nu} \delta q_j^\nu + \left. \frac{\partial \mathcal{J}}{\partial c_j} \right|_q \delta c_j \\ &= \tilde{q}_i^\mu \frac{\partial R_i^\mu}{\partial q_j^\nu} \delta q_j^\nu + \left. \frac{\partial \mathcal{J}}{\partial c_j} \right|_q \delta c_j \\ &= \left\{ \left. \frac{\partial \mathcal{J}}{\partial c_j} \right|_q - \tilde{q}_i^\mu \left. \frac{\partial R_i^\mu}{\partial c_j} \right|_q \right\} \delta c_j \end{aligned} \tag{35.31}$$

Note the use of Equation 35.29 (p. 2076) and Equation 35.30 (p. 2076) in the derivation of Equation 35.31 (p. 2076). The flow perturbation has now been eliminated from the expression, yielding a direct relation between the control variables and the observable of interest.

There are several important observations to be made about (Equation 35.30 (p. 2076)):

- The dimension of the problem to be solved is the same as the original flow problem, although the adjoint problem is linear.
- While the adjoint solution may be considered strictly as a vector of numeric values, experience with the continuous adjoint provides guidance on how the adjoint solution can be interpreted. The vector of weights associated with the components of the residual of the momentum equation in each cell is termed the *adjoint velocity*. The adjoint value associated with the residual of the continuity equation is termed the *adjoint pressure*.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The right hand side is defined purely on the basis of the observable that is of interest.
- The matrix on the left hand side is the *transpose* of the Jacobian of the governing system of equations (Equation 35.25 (p. 2075)). This seemingly innocent transposition has a very dramatic impact on how the adjoint system is solved. This will be discussed below.
- The adjoint equations are defined by the current state of the flow, and the specific physics that is employed in the modeling. Each adjoint solution is specific to the flow state.

At first glance it appears that solving the adjoint problem may be straightforward. After all it simply involves setting-up and solving a linear (albeit large) system of equations. In practice both steps can represent a significant challenge, especially when the problem is large.

The evaluation of the residuals of the flow equations is an integral part of the pre-existing ANSYS Fluent flow solvers. However, it is necessary to compute the Jacobian of the system $\partial R_i^\mu / \partial q_j^\nu$ and then transpose it, or at the very least to be able to make a matrix-free transpose matrix-vector product with the adjoint solution. There are several technical approaches to accomplishing this task whose description goes beyond the scope of this document. Suffice to say that it is not trivial to encode this functionality, but that it has been done successfully here.

For the present implementation, a pre-conditioned iterative scheme, based on pseudo-time marching, is adopted to solve the adjoint system. The equations that define the advancement process can be written as

$$\left\{\frac{P_{i,j}^{\mu,\nu}}{CFL\Delta t}+L_{ij}^{\mu,\nu}\right\}\Delta\tilde{q}_i^\nu=\frac{\partial \mathcal{J}}{\partial q_j^\mu}-\frac{\partial R_i^\nu}{\partial q_j^\mu}\tilde{q}_i^\nu \tag{35.32}$$

where Δt is a local time step size based on the local flow conditions, CFL is a user-specified CFL number. The matrix $L_{ij}^{\mu,\nu}$ is a simplified form of the system Jacobian that is amenable to solution using the AMG linear solver that is the workhorse of the conventional flow solver. The preconditioning matrix $P_{i,j}^{\mu,\nu}$ is a diagonal matrix. Artificial compressibility is introduced on the adjoint continuity equation to aid in the relaxation of the adjoint pressure field.

The correction $\Delta\tilde{q}_i^\mu$ is under-relaxed and added to the adjoint solution $\tilde{q}_i^\mu$ and the process repeated until the right hand side is adequately small.

35.1.5. Adjoint Solver Stabilization

When applied to problems with large cell counts and complex geometry, adjoint solvers sometimes experience stability issues. These instabilities can be associated with small scale unsteadiness in the flow field and/or strong shear, and tend to be restricted to small and isolated regions of the flow domain. Despite the spatial localization of these instabilities, the linearity of the adjoint problem provides no intrinsic limit on their growth during solution advancement. Their presence, if not handled, can disrupt the entire adjoint calculation despite the problem occurring in sometimes just a few cells. A stabilized solution scheme will be required to obtain adjoint solutions for problems at high Reynolds number in which there is strong shear and/or complex geometry.

Three stabilization schemes are available in Fluent in order to overcome these stability difficulties when larger cases are being solved. The stabilization schemes are designed to intervene only when the standard advancement scheme is experiencing instability.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Spatial scheme

The spatial scheme operates by identifying parts of the domain where unstable growth is occurring and applying a more direct and stable solution procedure in those regions.

Modal scheme

The modal scheme involves a process of identifying the particular details of the unstable growth patterns or modes. These patterns are localized in space and they are used to split the solution into parts that have stable and unstable characteristics when advanced. The stable part is advanced as usual, while the algorithm is designed now to compensate for the unstable part so that the overall calculation is stabilized.

The emergence of the unstable patterns can happen at any time during the adjoint calculation and the total number of unstable patterns is case-dependent. Having 10 to 20 unstable modes present would not be considered unusual. Large cases may have many more. As a general trend, the most rapidly-growing unstable patterns appear often within a few iterations with more slowly-growing modes appearing later in the calculation. The unstable patterns are handled as they appear.

Dissipation scheme

The dissipation scheme provides stabilization for the solution advancement of the adjoint solution by introducing nonlinear damping strategically into the calculation domain. The strategy is intended to provide minimal intervention in order to damp the growth of instabilities that lead to adjoint solution divergence. A marker is tracked, based on the state of the adjoint solution, and damping is applied directly to the adjoint solution in regions where the marker becomes large.

Unlike the modal and spatial schemes, the dissipation scheme can affect the adjoint solution slightly. In general, the spatial order of the damping is chosen to be one order larger than the adjoint calculation order. This means that the formal order of accuracy of the adjoint solver is unaffected by the addition of the dissipation scheme. In practice, regions of intense dissipation can appear for some cases which may result in isolated spots and/or streaks in the solution on surfaces. However, the scale of these solution features is such that they are often easily smoothed during postprocessing to generate design changes.

Some additional computational overhead is associated with the stabilization schemes. In general, the dissipation scheme is the least resource-intensive of the three schemes.

For information about using the stabilization schemes, refer to Stabilized Scheme Settings (p. 2094).

35.1.6. Solution-Based Adaption

An adjoint solution provides guidance on where best to adapt a computational mesh in order to resolve quantities of engineering interest.

Once the governing equations for the system (Equation 35.25 (p. 2075)) have been converged there remains a discretization error, ε_i^μ, such that

$$\mathcal{R}_i^\mu\left(\underline{q}^0,\underline{q}^1,\ldots,\underline{q}^{M-1};\underline{c}\right)=\varepsilon_i^\mu,\ \mu=0,\ldots,M-1,\quad i=0,\ldots,L-1. \tag{35.33}$$

While specific estimates for this discretization error may be tricky to define, it is often estimated to be $O\left(h^p\right)$, where h is the local grid size, and p is the order of the discretization scheme. That is, $p=1$ for a first-order scheme and $p=2$ for a second order scheme. Alternatively, ε_i^μ can be considered to be the residual associated with a solution that is not converged fully.

The correction to the flow field, δq_j^ν, that compensates for this inhomogeneity is given by

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\frac{\partial R_i^\mu}{\partial q_j^\nu}\delta q_j^\nu = -\varepsilon_i^\mu \tag{35.34}$$

from which it follows quickly that

$$\delta \mathcal{J} = \frac{\partial \mathcal{J}}{\partial q_j^\nu}\delta q_j^\nu = -\varepsilon_i^\mu \tilde{q}_i^\mu . \tag{35.35}$$

This simple expression provides an estimate of the effect of the presence of ε_i^μ on the observation, $\mathcal{J}$.

The presence of discretization errors, or lack of convergence, on the engineering quantity of interest is assessed by weighting the inhomogeneous term by the local adjoint solution. It is clear that even in regions of the domain where the residuals or discretization errors are small, an accompanying adjoint velocity or pressure that is large in magnitude implies that there may be a significant source of error in the observable. A finer mesh in regions where the adjoint is large will reduce the influence of discretization errors that may adversely affect the engineering result of interest. In practice, adapting cells which have large magnitude adjoint velocity and/or adjoint pressure will achieve this goal.

35.1.7. Using The Data To Improve A Design

Adjoint sensitivity data can be used to guide how to modify a system in order to improve the performance. The observable of interest can be made larger or smaller, depending upon the engineering goal.

A common strategy for deciding how to modify the system is based on the gradient algorithm. The underlying principle is quite simply that modifying a system in a manner to which it is most sensitive maximizes the effect of the change. The change to a control variable is made in proportion to the sensitivity of the value of interest with respect to that control variable.

Denote the sensitivity of the cost with respect to shape by

$$\delta \mathcal{J} = \frac{\partial \mathcal{J}}{\partial x_j^n}\delta x_j^n \tag{35.36}$$

where x_j^n is the j^{th} coordinate of the n^{th} node in the mesh. Here x_j^n is a notation for the subset of the control variables c_j for the system that correspond to mesh node positions. Then an adjustment

$$\delta x_j^n = \lambda \frac{\partial \mathcal{J}}{\partial x_j^n} \tag{35.37}$$

will provide the maximum adjustment to $\mathcal{J}$ for given L^2 norm of δx_j^n, where λ is an arbitrary scaling factor. Note that λ can be picked to be positive or negative depending upon whether $\mathcal{J}$ is to be increased or decreased respectively. This is essentially a statement of the method of steepest descent.

Furthermore, the change is estimated to first order to be

$$\delta \mathcal{J} = \lambda \frac{\partial \mathcal{J}}{\partial x_j^n}\frac{\partial \mathcal{J}}{\partial x_j^n} . \tag{35.38}$$

For a sufficiently small adjustment, the change to the observation will strictly have the same sign as the scaling factor λ, provided the gradient is not identically zero.

In regions where the sensitivity is high, small adjustments to the shape will have a large effect on the observable. This corresponds directly with the idea of engineering robustness. If a configuration shows high sensitivities, then the performance will likely be subject to large performance variations if there

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

are manufacturing inconsistencies. For a robust design, the goal is to have the sensitivity be tolerably small.

In practical cases the field $\partial \mathcal{J} / \partial x_j^\eta$ can be noisy, especially for large, turbulent flow problems. If the noisy field is used directly to modify a boundary shape using (Equation 35.37 (p. 2079)), then the modified surface can have many inflections. This is not helpful for engineering design work. In the next section, the use of mesh morphing technology not only to smooth the sensitivity field, but also to provide smooth boundary and interior mesh deformation, is described.

35.1.7.1. Smoothing and Mesh Morphing

As has been noted, for typical engineering problems, the shape sensitivity field can have smoothness properties that are not adequate to define a shape modification. Mesh morphing technology is used here for both two- and three-dimensional systems in a two-fold role. The first role is as a smoother for the surface sensitivity field. The second role is to provide smooth distortions not only of the boundary mesh, but also the interior mesh. This approach is very appealing since it functions for arbitrary mesh cell types.

Note

For simplicity, the smoothing and morphing equations are presented here for the case of a 2D domain. The same approach is extended and used for 3D problems.

A Cartesian or cylindrical region is defined such that it encompasses all or a part of the problem domain. Only the mesh nodes that fall within this region may move as a result of the morphing operation. A local (u,v) coordinate system is defined, where $u \in [0,1]$ and $v \in [0,1]$. A regular array of $N_u \times N_v$ control points is then distributed in the control volume. The motion of the mesh at any point in the domain is determined by the movement of the control points.

In general, an optimization problem may include both free-form minimization or maximization of the observable(s) and constraints that are localized in space. This requires an approach that can provide a deformation field that is well-behaved and consistent with manufacturability requirements, while still allowing locally sharper deformations where required to satisfy the imposed constraints.

To achieve this, two coincident sets of control points are created. Bernstein polynomials and B-splines are used, respectively, to map the control point motions to the computational mesh nodes.

The i^{th} Bernstein polynomial of degree ℓ is:

$$B_{i,\ell}(u) = \binom{\ell}{i} u^i (1-u)^{n-i} \tag{35.39}$$

The i^{th} B-spline of degree p is the non-periodic B-spline that uses a knot-vector, $\{t_i\}$, where:

$$t_i = \begin{cases} 0 & i \le p \\ (i-p)/(\ell-2p) & p < i < \ell - p \\ 1 & i \ge \ell - p \end{cases} \tag{35.40}$$

The value of the B-spline is defined recursively as:

$$S_{i,0}(t) = \begin{cases} 1 & t_i < t < t_{i+1}; t_i < t_{i+1} \\ 0 & \text{otherwise} \end{cases}$$
$$S_{i,j}(t) = \frac{t - t_i}{t_{i+j} - t_i} S_{i,j-1}(t) + \frac{t_{i+j+1} - t}{t_{i+j+1} - t_{i+1}} S_{i+1,j-1}(t) \tag{35.41}$$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Depending on the degree of the B-spline, the function is zero on some portion of the unit interval. This is in contrast to the Bernstein polynomials that are strictly non-zero everywhere on the unit interval.

Let x_i^v denote the i^{th} coordinate of the v^{th} mesh node, and let Δx_i^v denote the change in the position of the node due to the morphing process. Let $\Delta \eta_{jk}^i$ denote the displacement of the $(j,k)^{\text{th}}$ control point associated with the Bernstein polynomials in the i^{th} coordinate direction, and let $\Delta \zeta_{jk}^i$ denote the displacement of the $(j,k)^{\text{th}}$ control point associated with the B-splines. If (u^v, v^v) denotes the position of the v^{th} mesh node, then the displacement of the node is defined by the superposition:

$$\Delta x_i^v = \sum_{j=0}^{N_u} \sum_{k=0}^{N_v} \left(\Delta \eta_{jk}^i B_{j,\ell}(u^v) B_{k,m}(v^v) + \Delta \zeta_{jk}^i S_{j,\ell}(u^v) S_{k,m}(v^v) \right) \tag{35.42}$$

The control-point movement for the Bernstein polynomials controls the large-scale smooth deformation, while the control-point movement for the B-splines controls fine-scale motions.

35.2. Using the Adjoint Solver

This section describes the process for loading the adjoint solver module, as well as setting up, running, and postprocessing the adjoint solutions. Also, this section demonstrates the shape modification process that is guided by the adjoint solution.

The typical use of the adjoint solver involves the following steps:

1. Load or compute a conventional flow solution.
2. Specify the observable of interest.
3. Set the adjoint solver controls.
4. Set the adjoint solver monitors and convergence criteria.
5. Initialize the adjoint solution and iterate to convergence.
6. Postprocess the adjoint solution to extract the sensitivity of the observable with respect to boundary condition settings.
7. Postprocess the adjoint solution to extract the sensitivity of the observable with respect to shape of the geometry.
8. Modify boundary shapes based on shape-sensitivity data and recompute the flow solution.

This section provides information about using the ANSYS Fluent adjoint solver in the following sections:

35.2.1. Model Considerations for Using Adjoint Solver
35.2.2. Defining Observables
35.2.3. Solving the Adjoint
35.2.4. Postprocessing of Adjoint Solutions
35.2.5. Modifying the Geometry Using the Design Tool

In addition, see the Fluent Tutorial Guide for more information about using the adjoint solver. (To access tutorials and their input files on the ANSYS Customer Portal, go to http://support.ansys.com/training.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.2.1. Model Considerations for Using Adjoint Solver

The current adjoint solver implementation provides basic adjoint solutions that accompany a conventionally-computed flow solution provided certain criteria are met.

35.2.1.1. Basic Assumptions and Consistency Checks
35.2.1.2. User-Defined Sources

35.2.1.1. Basic Assumptions and Consistency Checks

The adjoint solver is implemented on the following basis:

- The flow state is for a steady single-phase flow, or ideal gas, that is either laminar or turbulent.
- For turbulent flows, a *frozen turbulence* assumption is made, in which the effect of changes to the state of the turbulence is not taken into account when computing sensitivities.
- For turbulent flows, standard wall functions are employed on all walls.
- For moving (that is, non-inertial) reference frame problems, the single reference frame (SRF) and multiple reference frame (MRF) approaches are supported (as long as the absolute velocity formulation is used), while the mixing plane model (MPM) is not. Note that for the MRF approach, the adjoint solver does not support having a zone with a frame of motion that is relative to an already moving reference frame.
- Solid zones are supported, as long as they do not use a moving reference frame approach.
- The adjoint solver uses methods that are first order accurate in space by default. If desired, you can select second order accurate methods (see Using the Adjoint Solution Methods Dialog Box (p. 2091)).
- The boundary conditions are only of the following types:
  - Wall (not including thin walls or shells)
  - Velocity inlet
  - Mass-flow inlet
  - Pressure inlet
  - Pressure outlet
  - Symmetry
  - Rotational and translation periodic
- Constant and user-defined sources are supported for the momentum and energy equations; for details on the latter, see User-Defined Sources (p. 2083). Note that temperature dependency is not taken into account with the adjoint solver.

It is important to note that these requirements are not strict conditions for the conventional flow solver, but rather modeling limitations for the adjoint solver.

When the adjoint solver is initialized or an observable is evaluated, and before iterations are performed, a series of checks is performed to determine the suitability of the existing flow solution for analysis with the adjoint solver. Two types of message may appear:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- *Warning message:* There are certain physics models and boundary condition types that are not explicitly modeled in the adjoint solver, but their absence does not disallow you from proceeding with the calculation. In this case, a warning message will be printed that explains the nature of the inconsistency.

  ```
  Checking adjoint setup...
   -- Warning: Model is active but not included in adjoint calculation: P1 radiation model Done
  ```

  The adjoint solver will still run in this case but the quality of the adjoint solution data can be expected to be poorer as a result of the inconsistency. This is because in such cases the unsupported settings will revert to corresponding supported settings for the adjoint solver. Though the calculation will proceed, it is important to note that the results produced should be considered on this basis. When reverting back to the fluid calculation, the original settings will be preserved and the modified settings are not migrated onto the original case. The warning indicates that the setting change will be made automatically and specifically for the adjoint solution.

- *Error message:* There are some model and boundary conditions that are incompatible with the computation of an adjoint solution with the current adjoint solver implementation. In this case, an explanatory error message will be printed in the console window.

  ```
  Checking adjoint setup...
   ** Unable to proceed: Model is active but not compatible with adjoint calculation:
    Transition SST turbulence model
  Done
  ```

  The adjoint solver will not run in this case and changes must be made manually to the settings identified in the console before the solver can be run.

35.2.1.2. User-Defined Sources

User-defined sources for the momentum and/or energy equations that are defined in compiled UDFs can be accounted for in the application of the adjoint solver, provided that the source derivatives with respect to flow variables and cell centroid coordinates are provided. This is achieved by creating the source term UDF definition in the usual fashion, with the exception that a special UDF macro is used, `DEFINE_SOURCE_AE` (The suffix _AE denotes Adjoint Enabled).

`DEFINE_SOURCE_AE` (`name`, `c`, `t`, `dS`, `eqn`)

Argument Type	Description
`symbol name`	UDF name
`cell_t c`	Index that identifies cell on which the source term is to be applied
`Thread *t`	Pointer to cell thread
`real dS[]`	Array that contains the derivative of the source term with respect to the dependent variable of the transport equation, as well as the flow variables and cell centroid position
`int eqn`	Equation number

Function returns

`real`

Consider a source that may depend on velocity, pressure, and cell-centroid coordinates:

$$S\left(u,v,w,p,x_c,y_c,z_c\right)$$

where u, v, and w are the Cartesian velocity components in the cell, p is the pressure in the cell, and $\left(x_c,y_c,z_c\right)$ are the cell centroid coordinates as defined by the `C_CENTROID` macro. If the source strength depends on one or more of these quantities, then the derivatives of the source strength with respect to those variables must be computed and filled in as entries to the `dS[]` array. In particular,

$$\mathrm{dS[D_U]}=\frac{\partial S}{\partial u}$$
$$\mathrm{dS[D_V]}=\frac{\partial S}{\partial v}$$
$$\mathrm{dS[D_W]}=\frac{\partial S}{\partial w}$$
$$\mathrm{dS[D_P]}=\frac{\partial S}{\partial p}$$
$$\mathrm{dS[D_X_CENTROID]}=\frac{\partial S}{\partial x_c}$$
$$\mathrm{dS[D_Y_CENTROID]}=\frac{\partial S}{\partial y_c}$$
$$\mathrm{dS[D_Z_CENTROID]}=\frac{\partial S}{\partial z_c}$$

must be defined in the UDF if there is a dependency of the source upon them. The specification of this derivative information is essential for sensitivities, including shape sensitivity, to be computed correctly by the adjoint solver.

35.2.2. Defining Observables

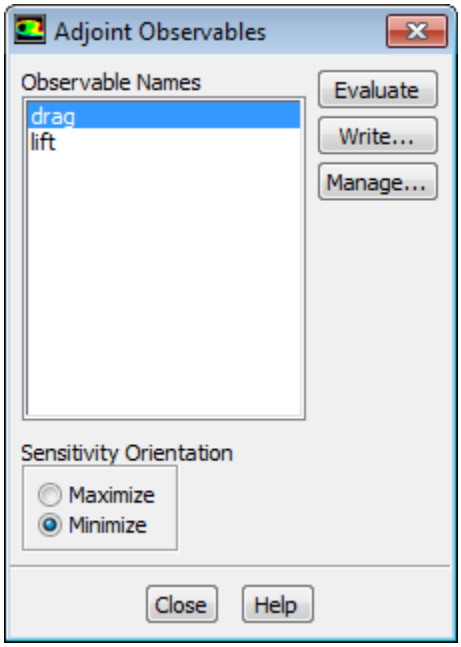
Defining an observable is a key initial step that must be performed for any adjoint calculation. The observable is the quantity for which sensitivities are sought. Observables can be defined based on flow variables, or as operations on other observables. For a list of the types of observables you can define refer to General Observables (p. 2069) and General Operations (p. 2072). You can define multiple observables, but only one observable at a time can be selected for the adjoint sensitivity calculation.

Observables are created and selected in the **Adjoint Observables** dialog box, which is accessed by clicking **Observable...** in the **Design** ribbon tab (**Adjoint-Based** group box). (Figure 35.1: Adjoint Observables Dialog Box (p. 2085)).

Design → **Adjoint-Based** → **Observable...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 35.1: Adjoint Observables Dialog Box



For more information, see the following sections:

35.2.2.1. Creating New Observables
35.2.2.2. Editing Observable Definitions
35.2.2.3. Selecting an Observable for Sensitivity Calculation

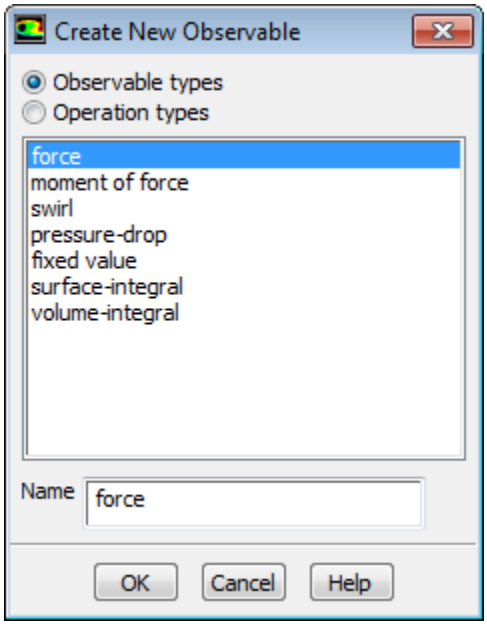
35.2.2.1. Creating New Observables

You can create a new observable by clicking **Manage...** in the **Adjoint Observables** dialog box and then **Create...** in the **Manage Adjoint Observables** dialog box.

In the **Create New Observable** dialog box (Figure 35.2: Create New Observable Dialog Box (Observable Types) (p. 2086)) you can choose from several observable types and operation types. Various types of observable quantities can be defined and each can be given a name. The available types of observables are described in General Observables (p. 2069).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

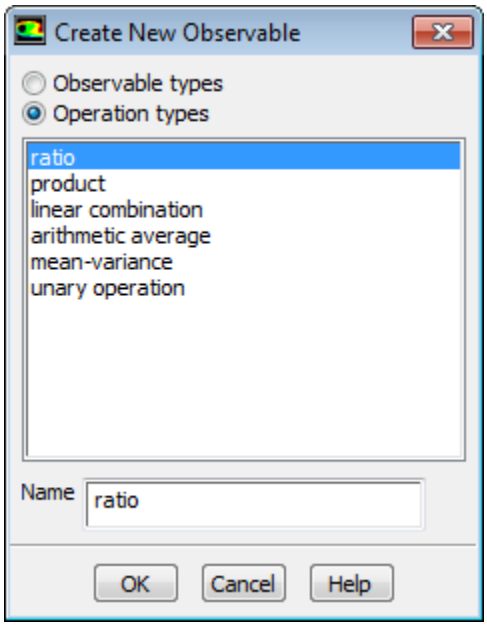
Figure 35.2: Create New Observable Dialog Box (Observable Types)



Select **Observable types** and pick an observable from the list. Keep the default name, or use the **Name** field to designate a different name for the observable of interest. Click **OK** to create the new observable, or click **Cancel** to dismiss the dialog box without creating an observable.

You can also combine existing observables in various ways or apply unary operations to create a wide variety of compound observables. The operations that you can apply to existing observables are described in General Operations (p. 2072).

Figure 35.3: Create New Observable Dialog Box (Operation Types)



Select **Operation types** and choose an operation from the list. Keep the default name, or use the **Name** field to designate a different name for the observable of interest. Click **OK** to create the new observable, or click **Cancel** to dismiss the dialog box without creating an observable.

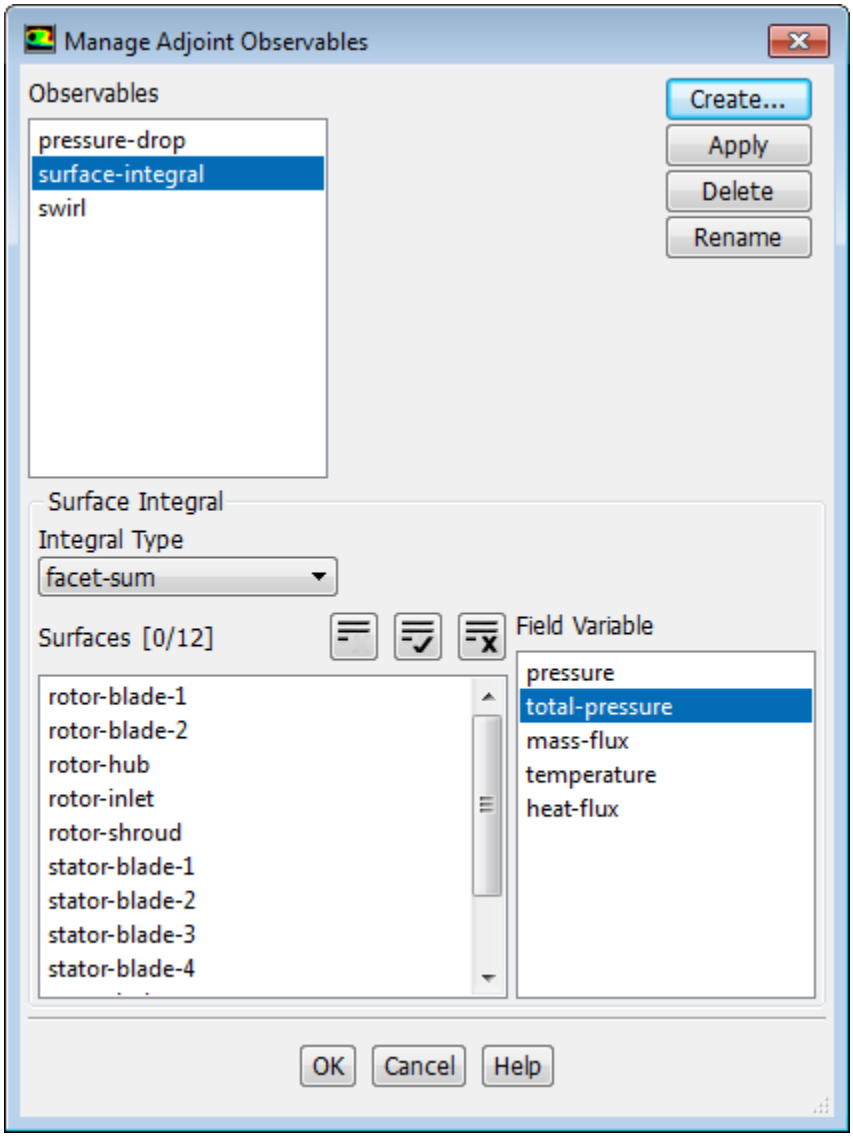
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Once created, you must edit the definition of the new observable (Editing Observable Definitions (p. 2087)).

35.2.2.2. Editing Observable Definitions

Once an observable is created, it appears in the **Observables** list. When you select an observable in the **Observables** list, the **Manage Adjoint Observables** dialog box changes to expose various properties that can be assigned for the selected observable.

Figure 35.4: Manage Adjoint Observables Dialog Box



Once the observable properties are defined (Inputs for Observable Types (p. 2088)), click **Apply** to apply the settings and proceed to define other observables, or click **OK** in the **Manage Adjoint Observables** dialog box to apply the settings and close the dialog box. Available observables are described in General Observables (p. 2069).

Note

It is permissible to leave observable definitions incomplete. The **Manage Adjoint Observables** dialog box can be considered as a workspace for the definition and manipu-

lation of observables. Only those operations with complete definitions will appear in the **Adjoint Observables** dialog box. Operations with undefined fields, or operations that depend on themselves, will be excluded from the list of usable observables.

Inputs for Observable Types

The inputs for each type of observable are summarized below:

- For a **force** observable:
  1. Select the walls that are to contribute to the force of interest in the **Wall Zones** list.
  2. Define the direction in which the force is to be computed by entering the components of this direction in the **X Component**, **Y Component**, and **Z Component** (for 3D) fields.
- For a **moment of force** observable:
  1. Select the **X**, **Y**, and **Z** (for 3D) components of the **Moment Center**.
  2. Select the **X**, **Y**, and **Z** (for 3D) components of the **Moment Axis**.
  3. Select a wall in the **Wall Zones** list.
- For a **swirl** observable:
  1. Select the **X**, **Y**, and **Z** (for 3D) components of the **Swirl Center**.
  2. Select the **X**, **Y**, and **Z** (for 3D) components of the **Swirl Axis**.
  3. Select a fluid in the **Fluid Zones** list.
- For a **pressure-drop** observable:
  1. Select the inlets and outlets between which the total pressure drop is to be computed using the **Inlets** and **Outlets** lists.
- For a **fixed value** observable, perform the following:
  1. Select the **Value** of the **Constant**.
- For a **surface-integral** observable:
  1. Select the type of integral from the **Integral Type** drop-down list. Available integral types are described in General Observables (p. 2069).
  2. Select a **Surface** and the corresponding **Field Variable**.
- For a **volume-integral** observable:
  1. Select the type of integral from the **Volume Integral Type** drop-down list. Available integral types are described in General Observables (p. 2069).
  2. Select the **Field Variable** of interest. If the chosen field variable is a vector quantity, you will need to specify a **Direction** along which the field variable will be evaluated.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Under **Zones of Integration**, select whether to integrate over selected **Zones** or **Integrate over a box**.

4. Select the **Zones** for integration under **Zone Settings** or the coordinates of the rectangular (in 2D) or hexahedral (in 3D) integration region under **Box Settings**.

- For a **ratio** observable:

  1. Select an observable from the **Numerator** drop-down list to represent the numerator of the ratio.

  2. Select an observable from the **Denominator** drop-down list to represent the denominator of the ratio.

- For a **product** observable:

  1. Select two observables from the corresponding drop-down list that will be used to compute their product.

- For a **linear combination** observable:

  1. Under **Linear Combination of powers**, select a **Constant** value.

  2. Select the number of **Components**.

  3. For each component, select a corresponding **Coefficient**, an **Observable**, and a **Power**.

- For an **arithmetic average** observable:

  1. Select the number of **Components**.

  2. Select observables from the corresponding **Observable** lists.

- For a **mean variance** observable:

  1. Select the number of **Components**.

  2. Select observables from the corresponding **Observable** lists.

- For a **unary operation** observable:

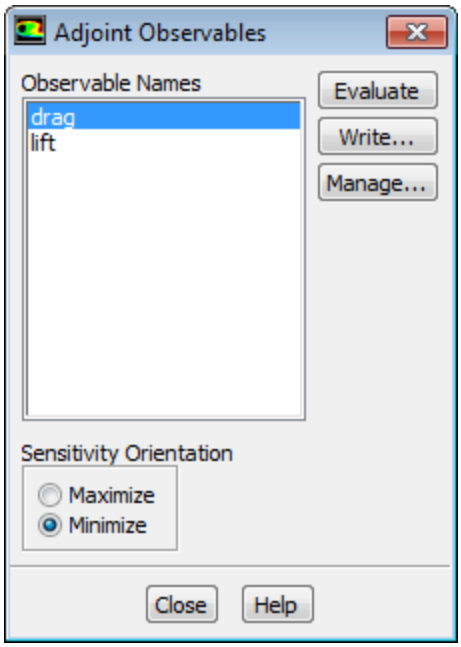
  1. Select an operation from the drop-down list (see General Observables (p. 2069) for a list of available operators).

  2. Select an observable from the corresponding drop-down list upon which the unary operation is to be applied.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.2.2.3. Selecting an Observable for Sensitivity Calculation

Figure 35.5: Adjoint Observables Dialog Box



You can select which observable to use for the sensitivity calculation in the list of **Observable Names**.

Note

- Only one observable can be used in any one adjoint calculation. In the **Adjoint Observables** dialog box, the observable that is currently highlighted is the observable that will be used in the sensitivity calculation.
- Any observables that have been created but which are missing required inputs will not appear in the list of **Observable Names**.

Clicking the **Evaluate** button computes the current value of the selected observable quantity and prints the result in the console window. Clicking the **Write...** button provides the option to write the result to a named file.

The **Sensitivity Orientation** determines the sign of the postprocessed sensitivities. For instance, if you select **Maximize** then changes in the direction of positive sensitivities will increase the value of the observable. A standard rule of thumb can be applied: if you want to improve the solution, then follow the direction of the sensitivity vectors. Another standard rule of thumb can also be applied for postprocessed scalars: if you want to improve the solution, then increase values where the sensitivity is positive and/or decrease values where the sensitivity is negative.

35.2.3. Solving the Adjoint

Once an observable is defined as described in Defining Observables (p. 2084), the solution of the adjoint for that observable can be computed based on the current flow solution. The steps for solving the adjoint are similar to those for solving the flow and are detailed in the following sections:

35.2.3.1. Using the Adjoint Solution Methods Dialog Box

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

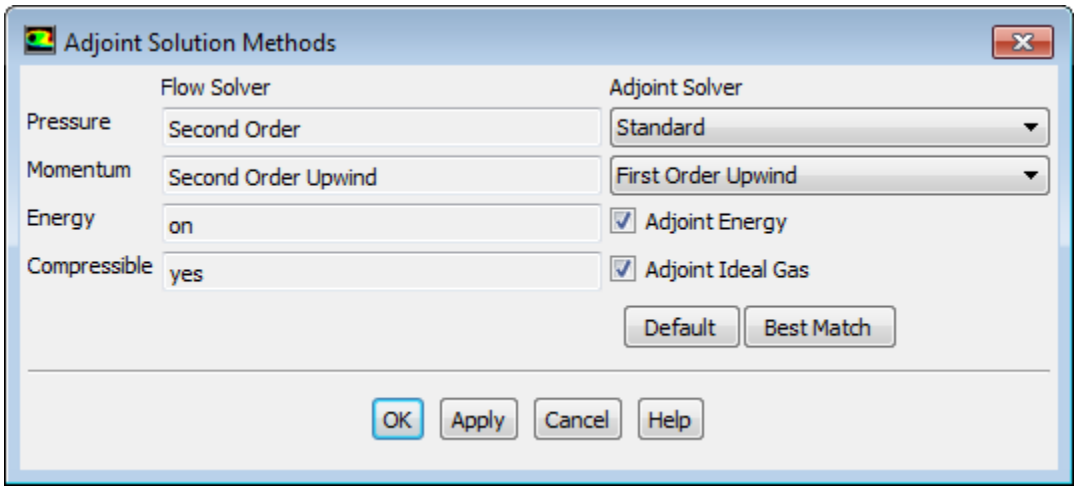
35.2.3.2. Using the Adjoint Solution Controls Dialog Box
35.2.3.3. Working with Adjoint Residual Monitors
35.2.3.4. Running the Adjoint Calculation

35.2.3.1. Using the Adjoint Solution Methods Dialog Box

You can specify the methods used for computing the adjoint solutions in the **Adjoint Solution Methods** dialog box. To open the **Adjoint Solution Methods** dialog box, click **Methods...** in the **Design** ribbon tab (**Adjoint-Based** group box).

Design → Adjoint-Based → Methods...

Figure 35.6: Adjoint Solution Methods Dialog Box



The **Adjoint Solution Methods** dialog box shows a side-by-side comparison of the schemes used for the flow solver and for the adjoint solver. Using the same scheme for the adjoint solution and the flow solution yields the most accurate discrete derivative calculation when the adjoint solution is converged. However, not all schemes used for the flow solver are supported for the adjoint solver. In these cases an alternate adjoint scheme must be used. This does not typically lead to severe deterioration of the adjoint results quality. Even if the same scheme is available for the adjoint solver this is not always practical because stability may be reduced with some schemes. Therefore, you can use the drop-down lists under **Adjoint Solver** to select alternate schemes as needed.

In some cases it may not be desirable to solve the energy adjoint even if energy is solved in the flow solver. For example, consider an incompressible flow where the specified observable does not involve thermal quantities. In this case the adjoint for the energy equation is identically zero, but its inclusion would add an unnecessary numerical burden. You can disable the **Adjoint Energy** option to avoid solving the energy adjoint.

If the primary flow is a compressible ideal gas, you can specify that the physics of the compressible flow are modeled in the adjoint solver by enabling the **Adjoint Ideal Gas** option. Enabling this option automatically enables the **Adjoint Energy** option as well. If the flow is not a compressible ideal gas, it will be treated as being incompressible for the purposes of the adjoint calculation.

Clicking **Default** will configure the adjoint solver to use the default low-order schemes (**Standard** for pressure and **First Order Upwind** for momentum), which are chosen for their stability. Clicking **Best Match** will attempt to match the adjoint schemes to the flow schemes that should in general provide more accurate calculations at convergence. Where they cannot be matched, the default scheme will be used.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.2.3.2. Using the Adjoint Solution Controls Dialog Box

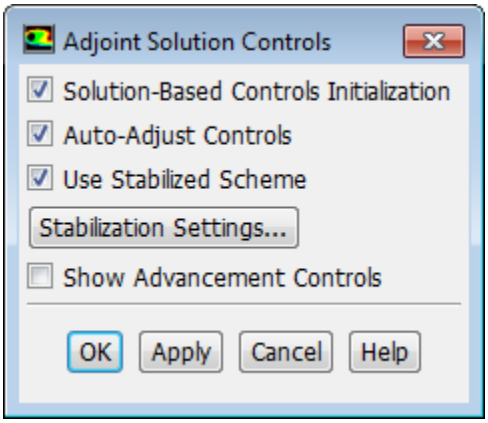
The advancement of the adjoint solution to reach a converged solution is an essential part of the analysis. The solution algorithm for the adjoint solver is similar to the coupled pressure-based solver that is available for conventional flow computations in ANSYS Fluent.

By default, the adjoint solver advancement settings will be automatically adjusted during calculation to encourage convergence, with initial values determined by the state of the flow solution. If needed, you can adjust the advancement controls in the **Adjoint Solution Controls** dialog box.

To open the **Adjoint Solution Controls** dialog box, click **Solver Controls...** in the **Design** ribbon tab (**Adjoint-Based** group box).

Design → **Adjoint-Based** → **Solver Controls...**

Options



Solution-Based Controls Initialization

When enabled, the advancement controls are selected automatically when the adjoint solution is initialized based on the state of the flow.

Auto-Adjust Controls

When enabled, the convergence of the AMG solver and the adjoint residuals are monitored during solution advancement. Based on the trends observed, the advancement controls are automatically adjusted to encourage reliable solution advancement and convergence. This is especially useful when a new type of problem is being solved for which appropriate settings are not initially clear. The advancement will begin with either user-specified settings or those from **Solution-Based Controls Initialization**. You can enable or disable **Auto-Adjust Controls** at any time during the solution.

If **Auto-Adjust Controls** is enabled, messages will appear as the calculation progresses indicating the adjustments that are being made.

Use Stabilized Scheme

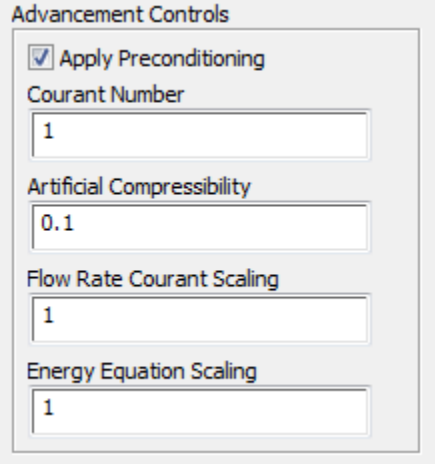
When enabled, stabilization will be applied to the adjoint solution. Click **Settings...** to specify the stabilization type and settings. The details of these settings are described in Stabilized Scheme Settings (p. 2094). A stabilized solution scheme will be required to obtain adjoint solutions for problems at high Reynolds number in which there is strong shear and/or complex geometry.

Show Advancement Controls

Toggles the visibility of the various advancement controls for direct user-specification.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Advancement Controls



Apply Preconditioning

Enables solution preconditioning. This is needed for most cases involving turbulent flow. If preconditioning is enabled, the following controls are available:

Courant Number

A higher number corresponds to a more aggressive advancement of the computation at the risk of instability. Disabling preconditioning corresponds to an infinite Courant number.

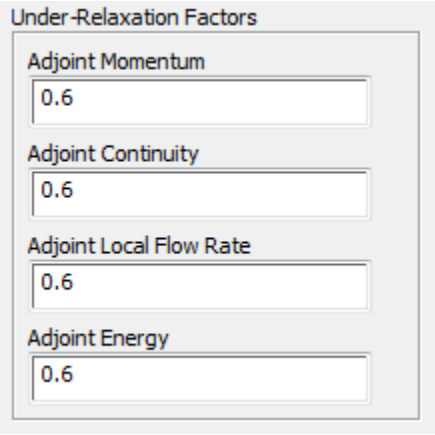
Artificial Compressibiilty

A nonzero value introduces artificial compressibility into the computation of the adjoint continuity equation. A value of `1.0` or less is reasonable.

Flow Rate Courant Scaling , Energy Equation Scaling

These values should be larger than zero (default values are `1`). A smaller value implies a less aggressive algorithm that encourages stability of the AMG linear solver.

Under-Relaxation Factors

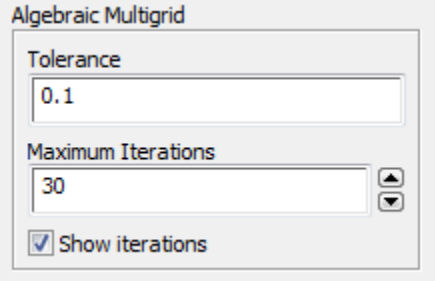


Adjoint Momentum , Adjoint Continuity , Adjoint Local Flow Rate , Adjoint Energy

Each of these can be set to a value between `0.0` to `1.0`. A higher value leads to a more aggressive algorithm that is less likely to be stable. A value of `1.0` for each can be used for some simple cases without difficulty.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Algebraic Multigrid



Tolerance
The tolerance used for judging convergence.

Maximum Iterations
The maximum number of inner iterations of the AMG solver.

Show Iterations
If enabled, a more verbose iteration history is printed in the text console during iterations. The details of the inner iteration can be useful when deciding on appropriate **Courant Number**, **Artificial Compressibility**, and **Flow Rate Courant Scaling**. If many inner iterations are needed, or if the inner iterations diverge, this signals that a reduction in **Courant Number** may be needed. Alternatively, an increase in **Artificial Compressibility** or a reduction in the **Flow Rate Courant Scaling** may be sufficient for the AMG iterations to converge.

35.2.3.2.1. Stabilized Scheme Settings

If you choose to use one of the stabilization schemes (Adjoint Solver Stabilization (p. 2077)) you will need to specify the scheme and settings in the **Stabilized Scheme Settings** dialog box (accessed by enabling **Use Stabilized Scheme** and clicking **Settings...** in the **Adjoint Solution Controls** dialog box).

The stabilization scheme settings are described in the following sections:
35.2.3.2.1.1. Modal Stabilization Scheme
35.2.3.2.1.2. Spatial Stabilization Scheme
35.2.3.2.1.3. Dissipation Scheme

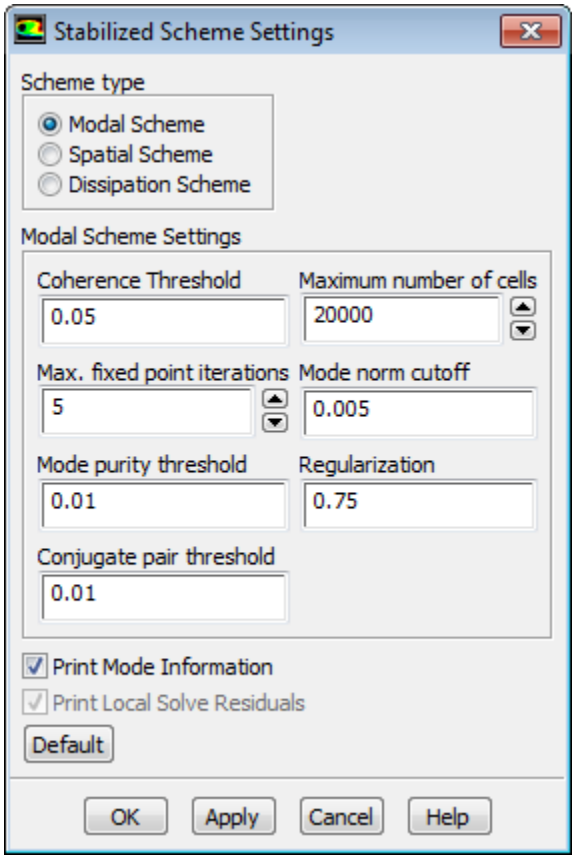
35.2.3.2.1.1. Modal Stabilization Scheme

The behavior of the modal scheme is as follows. As the calculation progresses the solution is monitored for unstable growing patterns or modes as the adjoint solution is corrected by the primary advancement algorithm. The first stage of the stabilization scheme involves a lightweight monitoring process to screen for any unstable patterns. If the calculation is progressing in a stable manner there is no indication even that the stabilization scheme is operating. If a candidate instability appears then it is monitored. In the event that unstable behavior that cannot be handled by the primary advancement scheme becomes evident a more intensive process is activated. Unstable patterns are refined using a fixed point iteration scheme so that they can be handled cleanly and advanced in a stable manner using an alternative to the primary advancement scheme.

The settings for the modal scheme are described below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 35.7: Stabilized Scheme Settings for Modal Scheme



Coherence Threshold

controls the preliminary screening for growth. It defines the threshold for relative variance of growth between iterations at which a more detailed search is activated. A smaller value will tend to delay the activation of the more detailed search for unstable growth at the risk of allowing unstable growth to continue for more iterations.

Max. Fixed Point Iterations

is the maximum number of iterations taken to refine the unstable pattern during the more detailed search.

Mode Purity Threshold

is the tolerance for an acceptable unstable pattern. A smaller value will lead to a more pure unstable mode being identified. This may occur at the expense of a larger number of fixed point iterations. If the purity threshold is not met after the maximum specified number of iterations then the unstable pattern is rejected.

Conjugate Pair Threshold

is the threshold for inclusion of a second unstable mode alongside a candidate mode, forming a conjugate pair. This corresponds to a pseudo-time-periodic instability, versus purely exponential growth. Such instabilities can occur in some cases. Increasing this value will increase the occurrence of a second mode being included.

Maximum Number of Cells

is the largest number of cells allowed for an unstable pattern that will be handled by the stabilization scheme. The problematic instabilities for an adjoint solver are often localized in space to a small number of cells out of the entire flow domain. However, there can be transiently growing corrections to the adjoint solution that masquerade as unstable patterns. This cell limit suppresses such false patterns from consideration.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mode Norm Cutoff
serves to identify the main body of an unstable pattern and exclude the peripheral spatial parts of the mode that are of small amplitude. If the norm cutoff were set to zero, all unstable modes would involve all cells in the domain. Therefore, reducing this value increases the number of cells that are included in an unstable mode.

Regularization
acts to regularize the advancement of the unstable mode components. The stiffness of the adjoint governing equations can give rise to difficulty in discerning the unstable pattern from nearby stable patterns. Increasing the regularization tends to overcome this difficulty and adds stability to the calculation at the expense of potentially slower convergence. Valid values are between `0` and `1`.

Print Mode Information
enables printing of stabilization information to the console.

When using the modal stabilization scheme, the following behavior can be observed:

- When the primary advancement scheme is proceeding in a stable manner, the stabilization scheme will remain idle and no output will be generated.

- When a candidate unstable pattern appears it will be tracked and a message will appear with information about the detected mode:

```
Mode growth:    1.34390e+00  Coherence:    8.28546e-02
```

  In the above example, the message indicates that an unstable pattern that grows by more than 34% in one iteration is present. The value of the Coherence corresponds to the variance in growth between iterations.

- When a candidate unstable pattern is identified that meets the coherence threshold, the secondary refinement stage is activated:

```
Mode growth:    1.35752e+00  Coherence:    3.95870e-02
Growing pattern cell count: 929
Isolating new mode
Iteration: 0  Growth rate:    1.42665e+00   Residual:    5.88196e-02
Iteration: 1  Growth rate:    1.41329e+00   Residual:    1.34600e-02
Iteration: 2  Growth rate:    1.41227e+00   Residual:    2.25348e-02
New growing mode with 597 cells
```

  If the growing pattern cell count exceeds **Maximum Number of Cells** the refinement procedure is postponed. Likewise, if after refinement the number of cells involved exceeds the specified limit the mode is rejected. Each iteration corresponds to a refinement of the unstable pattern. The adjoint solution itself is not advanced during this process. The residual corresponds to the purity of the mode and must eventually fall below the **Mode Purity Threshold** for the mode to be accepted. A minimum of 3 iterations is taken so that the occurrence or absence of a conjugate pair can be verified.

- The presence of the sometimes violent unstable growth that can occur gives a jagged appearance to the adjoint residual plots. However, the underlying trend of the residual plots should correspond to a converging adjoint solution process.

The unstable patterns of solution advancement are specific to the flow solution, choice of discretization schemes, and solution advancement controls for the adjoint solver. If any of these values changes, then the unstable modal patterns must be recomputed so that the stabilization scheme will function to full effect. In the event that any of these parameters is changed and the adjoint solution is restarted a mode recalibration process will be initiated. It should be noted that if there are many unstable modes that have been identified then this recalibration may take some time.

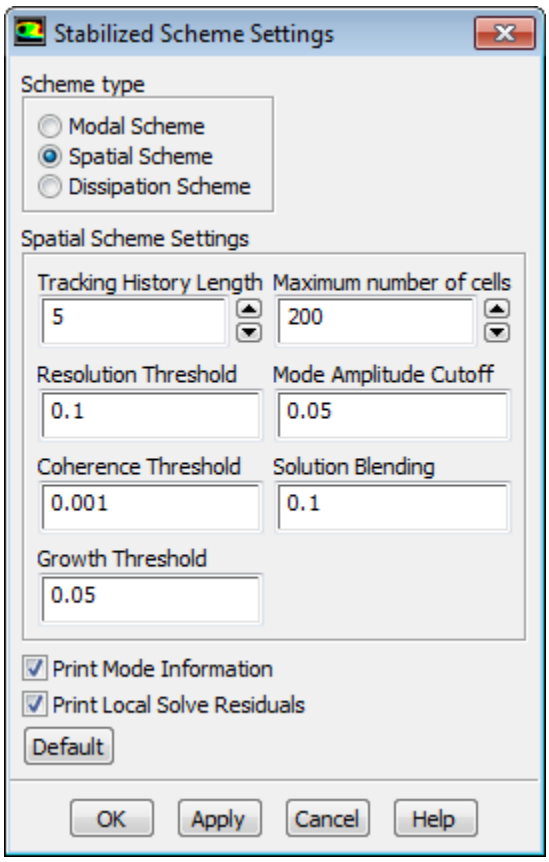
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In contrast, the patterns of unstable growth are not specific to the observable that has been selected. Given that there is computational effort needed to identify the instabilities, a separate initialization process is available just for the unstable modal patterns. This allows the adjoint solution itself to be initialized for a new observable while retaining the knowledge of how to stabilize the solution advancement. A much smoother path to solution for the second and subsequent observables is then achieved.

35.2.3.2.1.2. Spatial Stabilization Scheme

The spatial scheme operates by identifying parts of the domain where unstable growth is occurring and applying a more stable scheme. The unstable mode tracking and handling is done in an automated manner. Several settings are exposed, but are recommended for expert users only. When the stabilized scheme is used and the **Settings...** button is selected, the following dialog box will appear (Figure 35.8: Stabilized Scheme Settings for Spatial Scheme (p. 2097)):

Figure 35.8: Stabilized Scheme Settings for Spatial Scheme



The four items on the left side of the dialog box relate to unstable mode tracking and selection, while the items on the right provide options for the response once a mode is identified. The values have the following meaning:

Tracking History Length
: defines the number of iterations that are employed in the unstable mode identification process. A range between 2 and 5 may be chosen. The longer the tracking history, the better the quality of the identification process.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Resolution Threshold

defines a threshold for the separation between modes based on their amplitudes. A value of 0.1 implies that the second most-dominant mode must be 0.1 of the amplitude of the primary mode before an unstable mode can be considered as being adequately resolved.

Coherence Threshold

defines a threshold that must be met for the closeness of the mode to pure exponential growth as the iterations progress. A smaller number means that a tighter tolerance must be met before an unstable mode is considered as having been identified.

Growth Threshold

defines the growth rate threshold for unstable modes that must be met before the algorithm responds to the presence of the mode. The growth threshold defines the number of iterations that would have to take place for the mode to grow by one order of magnitude. That is, a value of 0.01 signifies that a mode that would grow one order of magnitude or more in 100 iterations will be handled. A negative value for this threshold is not recommended as multiple stable modes may be tracked.

Maximum Number of Cells

defines a limit on the number of cells for which stabilization will be applied when an unstable mode is located.

Mode Amplitude Cutoff

defines the fraction of the peak mode amplitude beyond which cells will be included for stabilization.

Solution Blending

defines an under-relaxation factor for blending the stabilized part of the scheme with the standard advancement scheme. A default of 0.1 is chosen. The value should lie between 0 and 1.

Print Mode Information

(when enabled) leads to details of the unstable mode tracking process being printed in the console window. The following is an example of what is printed when a mode is resolved partially:

```
Mode present> Growth :    2.38166e-01  Coherence  :    1.27858e+00
Resolution Threshold :    1.00000e-01  Resolution :    1.32611e-01
```

The following is an example of what is printed when a mode becomes more fully resolved:

```
 Mode present> Growth :    1.35318e-01  Coherence  :    2.84133e-04
 Resolution Threshold :    1.00000e-01  Resolution :    2.75312e-04
 Growth Threshold     :    5.60193e-02  Growth     :    1.35318e-01
 Coherence Threshold  :    1.00000e-02  Coherence  :    2.84133e-04
 Local stabilization required: Instability detected in 5 cells
```

The final line only appears when a coherent and resolved instability has been identified successfully. Subsequently, the solution advancement algorithm automatically adjusts to eliminate this unstable behavior.

Print Local Solve Residuals

(when enabled) leads to details of the residuals for the local solution process being printed in the console window. An example of what is printed is given below:

```
Iter   Adjoint-continuity  Adjoint-xMom     Adjoint-yMom      Adjoint-Flowrate
 ....
  27       1.76334e-03      1.57503e-01      1.26984e-01        2.92618e-01
Local (%)     61.02095         57.41937         62.09185           99.99136
```

The percentage residual indicates the fraction of the total residual that is associated with the local solution process.

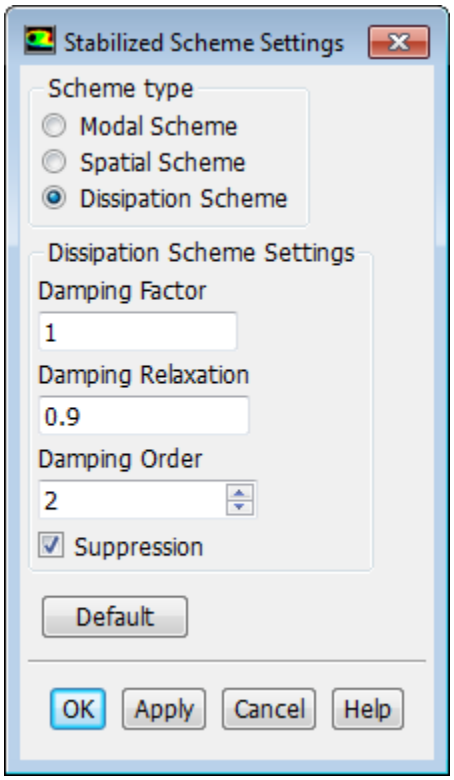
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Default
provides a set of default values for the various fields in the dialog box.

35.2.3.2.1.3. Dissipation Scheme

The dissipation scheme operates by introducing nonlinear damping strategically into the calculation domain. The strategy is intended to provide minimal intervention in order to damp the growth of instabilities that lead to adjoint solution divergence. A marker is tracked, based on the state of the adjoint solution, and the damping is applied directly to the adjoint solution in regions where the marker becomes large.

Figure 35.9: Stabilized Scheme Settings for Dissipation Scheme



The following parameters can be set:

Damping Factor
The overall level of dissipation is proportional to this value although the damping level is determined ultimately through a nonlinear process.

Damping Relaxation

The damping relaxation can be used to control the rate at which the dissipation is updated as the adjoint solution progresses. As you decrease the value from 1, the rate at which the dissipation is updated is decreased.

Damping Order
The spatial order of the dissipation. A higher order leads to more intense and localized damping. This can typically be set to be one order larger than the adjoint calculation spatial order.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Suppression

As part of the dissipation scheme, the growth of instabilities is stopped by the effect of the dissipation; enabling the **Suppression** option ensures that these undesirable patterns will then also decay as the calculation progresses.

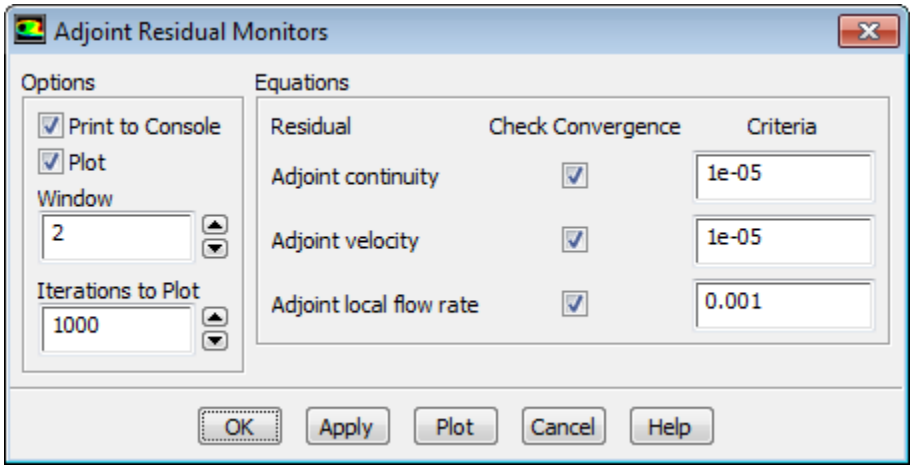
Since the order of discretization of the dissipation scheme is usually set higher than the adjoint, the formal order of accuracy of the adjoint solution is unaffected by the addition of dissipation. However, regions of intense dissipation can appear for some cases when using this scheme. This can sometimes appear as isolated spots and streaks in the adjoint solution on surfaces. The scale of these features is such that they are often easily smoothed during postprocessing to generated design changes.

35.2.3.3. Working with Adjoint Residual Monitors

The progress of the iterations of the adjoint solver and the monitoring of the convergence is controlled in the **Adjoint Residual Monitors** dialog box. This dialog box is accessed by clicking **Monitors...** in the **Design** ribbon tab (**Adjoint-Based** group box).

Design → Adjoint-Based → Monitors...

Figure 35.10: Adjoint Residual Monitors Dialog Box



The steps to define the monitor behavior are as follows:

1. Decide whether or not the residuals are to be printed in the console by setting the **Print to Console** check box as desired.

2. Decide whether or not the residuals are to be plotted in the main window by setting the **Plot** check box as desired.

3. Set the id of the window in which the adjoint residuals are to appear in the **Window** field.

4. Set how many iterations are to be shown in the residual curves that are plotted in the **Iterations to Plot** field.

5. Enable and/or disable those values that are to be used as criteria for convergence, and set the **Criteria** in each case. The options are to test the residuals for the **Adjoint continuity** and/or **Adjoint velocity** and/or **Adjoint local flow rate** equations.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

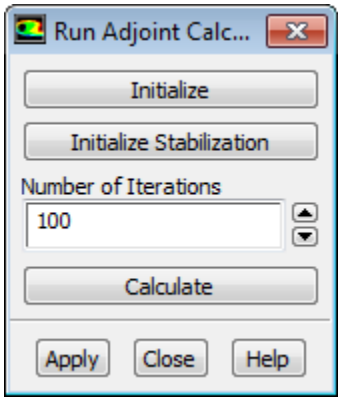
Click the **OK** or **Apply** buttons to confirm that the settings are acceptable. Clicking **Plot** will cause existing adjoint residuals to be plotted in the main graphics window.

35.2.3.4. Running the Adjoint Calculation

Initialization and execution of the adjoint solver is accomplished in the **Run Adjoint Calculation** dialog box. This dialog box is accessed by clicking **Calculate...** in the **Design** ribbon tab (**Adjoint-Based** group box).

Design → **Adjoint-Based** → **Calculate...**

Figure 35.11: Run Adjoint Calculation Dialog Box



The function of this dialog box is as follows:

Initialize
sets the value of the adjoint velocity and pressure to zero everywhere in the problem domain.

Initialize Stabilization
clears all of the unstable modes that have been identified in calculations using the modal stabilization scheme, but leaves the adjoint solution unchanged. Only available when **Modal** is selected in the **Stabilized Scheme Settings** dialog box.

Calculate
advances the adjoint solver by the **Number of Iterations** specified in the adjacent field.

Depending on the monitor settings, the residuals may be printed in the console and/or plotted in the main graphics window.

35.2.4. Postprocessing of Adjoint Solutions

Adjoint solution data can be postprocessed so that it provides both qualitative and quantitative views of the effect of many types of change that may be imposed on a system. While shape changes are often of particular interest there is a very rich data set, one as large as the original flow field, available for exploration. Postprocessing tools are provided with the intention of permitting the adjoint data to be mined in such a way that it provides useful supporting information for an engineer who is making design decisions for a system, or has questions about the reliability of the original flow calculation.

Since the ANSYS Fluent adjoint solver is a discrete adjoint solver, the primitive adjoint solution data provides sensitivity to changes cell-wise for the computational mesh. Normalization of these results by the cell volume provides a *mesh-independent* view into the data set.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Information regarding postprocessing the adjoint solutions can be found in the following sections:
35.2.4.1. Field Data
35.2.4.2. Scalar Data

35.2.4.1. Field Data

Adjoint solution data can be postprocessed using standard ANSYS Fluent postprocessing tools including contours, vectors, xy-plots, histograms, and surface and volume integrals.

In the **Contours** dialog box, under **Sensitivities...**, you can find the following fields:

Magnitude of Sensitivity to Body Forces (Cell Values)
This field is the magnitude of the adjoint velocity primitive field. This field can be interpreted at the cell level as the sensitivity of the observable to a body force on the cell, or any factors that may change the momentum balance on the cell. As such it is a mesh-dependent quantity. This field is often observed to be large, for example, upstream of a body for which drag sensitivity is of interest, with the field diminishing in the upstream direction. This indicates the interference effect for an object positioned at various locations upstream of the object of interest.

Sensitivity to Body Force X-Component (Cell Values), Sensitivity to Body Force Y-Component (Cell Values), and Sensitivity to Body Force Z-Component (Cell Values)
These are the components of the adjoint velocity primitive field. These fields show the sensitivity of the observable to the various components of a body force that may be applied to a cell, or other factors that may change the momentum balance on the cell. As such, it is a mesh-dependent quantity.

Sensitivity to Mass Sources (Cell Values)
This field provides a plot of the primitive adjoint pressure field. This field can be interpreted at the cell level as the sensitivity of the observable with respect to any changes to the mass balance on the cell. As such, it is a mesh-dependent quantity.

Sensitivity to Energy Sources (Cell Values)
This field is available when the energy adjoint is solved and is the primitive adjoint temperature field. It can be interpreted at the cell level as the sensitivity of the observable with respect to changes in the thermal energy balance of the cell. As such, it is a mesh-dependent quantity.

Magnitude of Sensitivity to Body Forces
This field is the magnitude of the adjoint velocity primitive field normalized by cell volume. This field can be interpreted as the magnitude of the sensitivity of the observable to body force per unit volume. It can be used to identify regions in the domain where small changes to the momentum balance in the flow can have a large or small effect on the observable. This field is often observed to be large, for example, upstream of a body for which drag sensitivity is of interest, with the field diminishing in the upstream direction. This indicates the interference effect for an object positioned at various locations upstream of the object of interest. This field is normalized so that, in principle, it shows a mesh-independent quantity. In practice, this field has features that are often correlated with mesh structure. This is not unexpected. In fact, the adjoint solution field is providing very useful insight into regions of the flow domain where the observation of interest is potentially sensitive to particular features of the mesh and discretization process that affect the local momentum balance.

Sensitivity to Body Force X-Component, Sensitivity to Body Force Y-Component, and Sensitivity to Body Force Z-Component (in 3D) fields
These are the components of the adjoint velocity primitive field normalized by cell volume. This field can be interpreted as the magnitude of the sensitivity of the observable to components of a body force per unit volume. Consider a body force distribution, expressed as a force per unit volume. The

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

integral of the vector product of that distribution with the components of this field gives a first-order estimate of the net effect of the body force on the observation.

Sensitivity to Mass Sources

This field is the primitive adjoint pressure field normalized by cell volume. This field can be interpreted as the sensitivity of the observable with respect to mass sources or sinks in the domain. Consider a mass source/sink distribution, expressed as mass flow rate per unit volume. The integral of that distribution, weighted by the local value of this field, gives the effect of the sources/sinks on the observation. This field is normalized so that, in principle, it shows a mesh-independent quantity. In practice, this field has features that are often correlated with mesh structure. This is not unexpected. In fact the adjoint solution field is providing very useful insight into regions of the flow domain where the observation of interest is potentially sensitive to particular features of the mesh and discretization process that affect local mass balance. When plotted on a boundary, this field indicates the effect of the addition or removal of fluid from the domain upon the quantity of interest. It is important to note that in this scenario the effect of the momentum of the fluid that is added or removed is not taken into account. The boundary velocity sensitivity should be plotted if that effect is also of interest.

Sensitivity to Energy Sources

This field is available when the energy adjoint is solved and is the primitive adjoint temperature field normalized by cell volume. It can be interpreted at the cell level as the sensitivity of the observable with respect to thermal energy sources or sinks in the domain. This field is normalized so that, in principle, it is a mesh-independent quantity.

Sensitivity to Viscosity

This field shows the sensitivity of the quantity of interest to variations in the turbulent effective viscosity for a turbulent problem, or the laminar viscosity in a laminar case. The sensitivity is normalized by the cell volume to account for cell size variations in the mesh.

Shape Sensitivity Magnitude

This field is the magnitude of the sensitivity of the observable with respect to a deformation applied to the mesh (both boundary and interior mesh). When plotted on the surface of a body the locations where this quantity is large indicates where small changes to the surface shape can have a large effect on the observable of interest. If the shape sensitivity magnitude is small then the effect of shape changes in this region can be expected to have a small effect on the observable of interest. When viewing this field, it is often observed that the magnitude varies by many orders of magnitude. Contour plots will clearly draw attention to regions with the highest sensitivity (often sharp edges and corners). However, it should be remembered that a relatively small surface movement that is distributed over a large area can have a cumulative effect that is large.

Normal Shape Sensitivity

This field shows the normal component of the shape sensitivity. A positive value indicates an orientation directed into the domain, while a negative value indicates that the shape sensitivity is oriented outwards from the domain. This field eliminates the component of the vector shape sensitivity field that lies in the plane of the wall.

Normal Optimal Displacement

This field shows the normal component of the optimal displacement computed from the adjoint solution. This field is defined only for portions of walls lying within the control-volume specified for morphing. The size of the optimal displacement is determined by the scale factor that is chosen for the morphing. A positive value of displacement indicates that the surface will be displaced into the flow domain, whereas a negative value of displacement corresponds to wall movement outwards from the flow domain. This field eliminates the component of the optimal displacement vector that lies in the plane of the wall.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

log10(Shape Sensitivity Magnitude)

In view of the large range of values possible for the shape sensitivity magnitude, a convenience function that plots $\log_{10}$ of the magnitude is provided. This allows the importance of the surfaces in a domain to be ranked more easily based on how they affect the observation of interest when they are reshaped.

Shape Sensitivity X-Component, Shape Sensitivity Y-Component, and Shape Sensitivity Z-Component (in 3D) fields

These fields are the individual components of the sensitivity of the observable of interest with respect to the mesh node locations. It is plotted as cell data and is computed as the average of the nodal sensitivities for a given cell, divided by the cell volume. Note that for this discrete adjoint solver the sensitivity of the result with respect to node locations both on and off boundaries is computed. The normalization by cell volume indicates that the fields that are plotted are the weighting factors for a continuous spatial deformation field. (Note that the nodal sensitivity data itself is used when mesh morphing is performed, and predictions about the effect of shape changes are made.)

Sensitivity to Boundary X-Velocity, Sensitivity to Boundary Y-Velocity, and Sensitivity to Boundary Z-Velocity (in 3D) fields

These fields are defined on those boundaries where a user-specification of a boundary velocity is made for the original flow calculation. This includes no-slip walls. The field shows how sensitive the observable of interest is to changes in the boundary velocity at any point. It is interesting to note that even though the original boundary condition specification may be for a uniform velocity on the domain boundary, the effect of a non-uniform velocity perturbation is available, The effect of any specific boundary velocity change can be estimated as an integral of the vector product of the change to the velocity with the plotted sensitivity field. A plot of this quantity on a velocity inlet, for example, can be very useful for assessing whether or not the inlet is positioned too close to key parts of the system. That is, it addresses the question of whether or not the flow domain is too small to achieve a successful computation of the performance measure of interest. Viewing this field will also indicate whether or not the assumption of a uniform inflow is adequate.

Sensitivity to Boundary Pressure

This field is defined on boundaries where there is a user-specified pressure as part of a boundary condition, such as on a pressure outlet. The field shows the sensitivity of the observation of interest to variations in the boundary pressure across the flow boundary. It is interesting to note that even though the original boundary condition specification may be for a uniform pressure on the domain boundary, the effect of a non-uniform pressure perturbation is available, The effect of any specific boundary pressure change can be estimated as an integral of the product of the change to the pressure with the plotted sensitivity field. Viewing this field will also indicate whether or not the assumption of a uniform pressure is adequate for the simulation.

Sensitivity to Boundary Temperature

This field is available when the energy adjoint equation is solved and is defined on boundaries where a temperature boundary condition is applied. This includes walls, velocity inlets, mass-flow inlets, pressure inlets, and pressure outlets where a backflow temperature may be specified. The field shows the sensitivity of the observation of interest to variations in the boundary temperature across the boundaries. Note that even if the original boundary condition specification is for a uniform temperature on the boundary, the effect of a non-uniform temperature perturbation is available. The effect of any specific boundary temperature change can be estimated as an integral of the product of the change to the temperature with the plotted sensitivity field. This field can be used to indicate whether or not the assumption of a uniform temperature is adequate for the simulation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sensitivity to Boundary Heat Flux

This field is available when the adjoint energy equation is solved and is defined on walls where a heat flux boundary condition is imposed. The field shows the sensitivity of the observation of interest to variations in the boundary heat flux through the wall. Its properties are analogous to those of Sensitivity to Boundary Temperature.

Sensitivity to Flow Blockage

This field is provided as a convenient tool for identifying portions of the flow domain where the introduction of blockages or obstructions in the flow can affect the observation of interest. Consider a blockage in the flow that generates a reaction force on the flow that is proportional to the local flow speed, and acting in the opposite direction to the local flow: $F=-\alpha(r)v$ where α is a local coefficient for the reaction force. The local contribution of this force on the observation of interest is determined by the vector product of this force with the adjoint velocity field. The flow blockage field that is plotted is $-v\cdot\tilde{v}$, namely the negative of the vector product of the flow velocity and the adjoint velocity (Cell Value).

Adjoint Local Solution Marker

This field, intended for expert users, can be plotted to identify those portions of the flow domain where the stabilized adjoint solution advancement scheme is applied. It is preferable to plot this with the **Node Values** de-selected in the **Contours** dialog box. In this case, the **Adjoint Local Solution Marker** will take a value between 0 and 1. The **Mode Amplitude Cutoff** defined in the **Stabilized Scheme Settings** dialog box defines the lower bound for cells where the stabilized scheme is applied.

The surface shape-sensitivity fields contain the derivatives of the chosen observable with respect to the position of the bounding walls of the problem.

In the **Vectors** dialog box, under **Vectors of...**, you can find the following custom vector fields:

Sensitivity to Body Forces (Cell Values)

This vector field shows how sensitive the observable of interest is to the presence of body forces within the flow. A body force that locally has a component in the direction of the body-force sensitivity vector will lead to an increase in the observation of interest. The larger the sensitivity vector, the larger the effect that a body force can have on the observation of interest.

Sensitivity to Body Forces

This vector field shows how sensitive the observable of interest is to the presence of body forces within the flow. It is normalized by the local cell volume so that it is *mesh-independent* data. A body force that locally has a component in the direction of the body-force sensitivity vector will lead to an increase in the observation of interest. The larger the sensitivity vector the larger the effect that a body force can have on the observation of interest. The first-order effect of a body force distribution per unit volume can be computed as a volume integral of the vector product of the body force distribution with this field.

Sensitivity to Shape field defined on walls

This vector field shows the postprocessed adjoint solution, independent of any morphing settings, that defines the sensitivity of the observable of interest to movement of the walls. Deformation of a wall where this vector is large can be expected to have a significant effect on the observable. In contrast, deformations of the wall in locations where this vector is small in magnitude, to first order, will have no effect on the observable.

Optimal Displacement

The optimal displacement field is defined once a design change has been computed using the design tool. This vector field shows the optimal displacement based on the design goals that have been specified and any prescribed boundary motions. Note that when plotting this vector field, if the **Auto Scale** option is

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

deselected, and the vector **Scale** option is set to 1, then the vectors as shown display the absolute displacement that will occur.

Sensitivity to Boundary Velocity

This sensitivity field is defined on boundaries where velocities are specified as an input for the boundary condition. This includes no-slip, and slip walls. When plotted, this field illustrates where a change to the boundary velocity can affect the observation of interest. When plotted on a wall, it shows how the imposition of a nonzero velocity condition on the wall will affect the observation. This can include movement in the plane of the wall itself. When plotted, for example on a velocity inlet, this field illustrates how local adjustments to the velocity of the incoming flow affect the observation of interest. Integrating a potential change to the velocity field, weighted by this sensitivity field over the boundary provides a first order estimate of the effect of the change on the observation of interest.

Sensitivity to Boundary Mass Flux

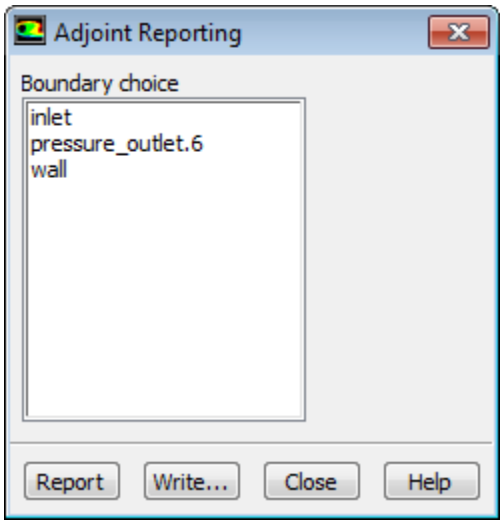
This sensitivity field that can be used to visualize the effect of adding or removing fluid via a wall-normal jet. This can be used, for example, to identify regions on walls where the introduction of suction could have a significant effect on the observable that has been specified. A plot of the **Sensitivity to Boundary Velocity** in general provides a richer view, but sometimes at the expense of the clarity of the visualization.

35.2.4.2. Scalar Data

The computed adjoint solution can provide sensitivity information for individual boundary condition settings through the **Adjoint Reporting** dialog box. This dialog box is accessed by clicking **Reporting...** in the **Design** ribbon tab (**Adjoint-Based** group box).

Design → Adjoint-Based → Reporting...

Figure 35.12: Adjoint Reporting Dialog Box



Boundary condition sensitivity data is retrieved as follows:

- Select a single boundary of the problem in the **Boundary choice** selection box.
- Click **Report** to see the boundary condition settings and the sensitivity of the observable with respect to those settings. An example of a report is shown below:

```
Velocity inlet id 5
Velocity magnitude = 4.000000000e+01 (m/s),
Sensitivity = 5.429127866e+01 ((n)/(m/s))
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In this case, the report shows that for every change of the boundary velocity by 1 m/s, a change in the predicted observable (in this case a force) of 54.2 N is expected. This prediction is based on a linear extrapolation.

- Click **Write...** to write the report for the selected boundary to a file.

35.2.5. Modifying the Geometry Using the Design Tool

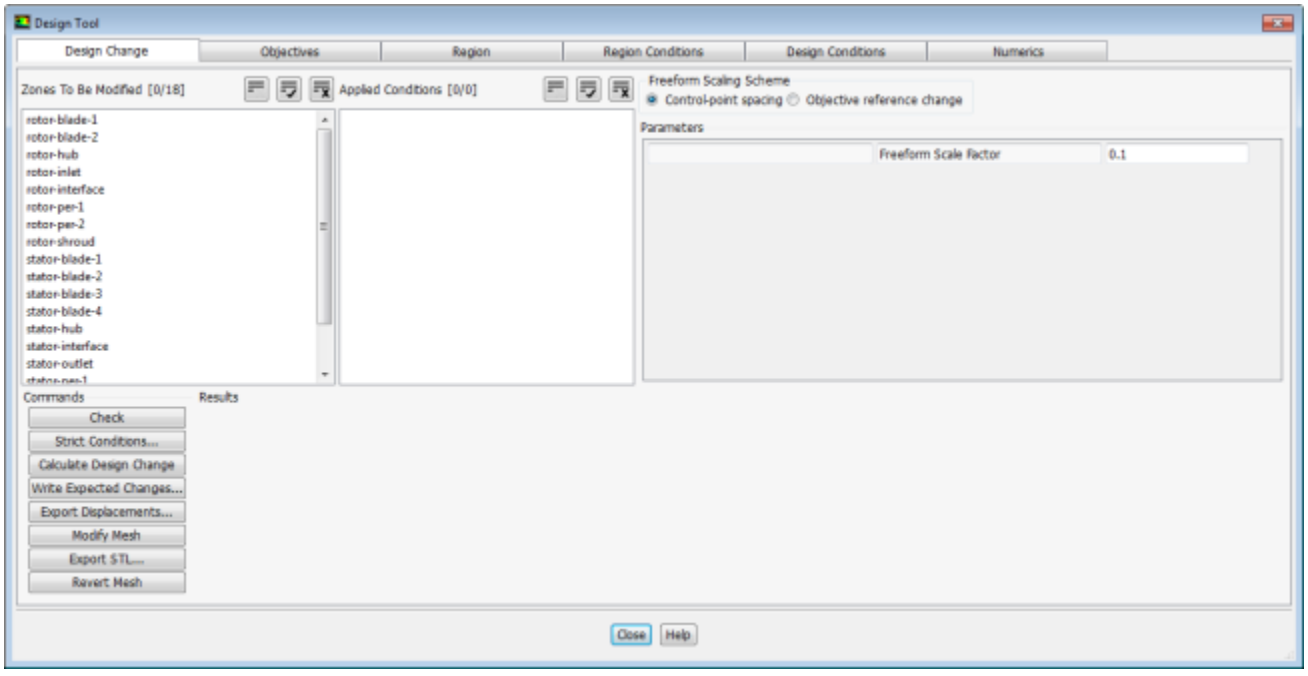
Tools to modify the boundary and interior mesh of the problem based on the adjoint solution are provided in the **Design Tool** dialog box. The **Design Tool** uses computed adjoint sensitivity data to find an optimal design change satisfying one or more goals or constraints, and to predict the resulting changes in observable values. Among the goals that can be specified in the **Design Tool** are:

- minimization, maximization, or targeted changes of single or multiple observable values
- prescribed deformations of selected geometry
- general constraints on deformations including restrictions on direction of motion, symmetry, and boundary conditions.
- equality and/or inequality constraints on geometry (fixed walls, bounding boxes, and so on).

The **Design Tool** is accessed by clicking **Design Tool...** in the **Design** ribbon tab (**Adjoint-Based** group box).

Design → Adjoint-Based → Design Tool...

Figure 35.13: Design Tool Dialog Box



The general procedure to use the Design Tool is as follows.

1. Select those boundaries that are free to be deformed in the **Zones To Be Modified** field on the **Design Change** tab. Boundaries that are not selected in this box that intersect the control volume will be constrained not to move. This is useful if there are walls which, for design reasons, must

not move. Note that if you have a two-sided wall that you want to deform, it is recommended that you select both the wall and the corresponding wall-shadow.

2. Define the region that will be modified in the **Region** tab. (Defining the Region for the Design Change (p. 2108))

3. Define conditions on the deformation region such as number of control points, symmetry, directional invariance, and boundary continuity. (Defining Region Conditions (p. 2110))

4. Compute and export the sensitivities from the adjoint solution for each observable of interest. (Exporting Sensitivity Data (p. 2111))

5. Import the sensitivity data for each observable that will participate in the design change and specify the optimization goals for the observables. (Defining Observable Objectives (p. 2112))

Note

Export and Import of sensitivity data is only required for multi-objective analysis.

---

6. Define constraining and/or deformation conditions on the design change, such as fixed boundaries, prescribed boundary deformation (through a profile, translation, rotation, or scaling), and bounding surfaces / planes. (Defining Conditions for the Deformation (p. 2112))

7. Specify weights for the various objectives, a scaling factor for the deformation, and whether the design conditions are strictly enforced. (Shape Modification (p. 2117))

8. (optional) Adjust the optimal design calculation numerics if required.

9. Compute the optimal design change and modify the mesh.

Details of these steps are elaborated in the following sections:

35.2.5.1. Defining the Region for the Design Change
35.2.5.2. Defining Region Conditions
35.2.5.3. Exporting Sensitivity Data
35.2.5.4. Defining Observable Objectives
35.2.5.5. Defining Conditions for the Deformation
35.2.5.6. Shape Modification
35.2.5.7. Design Tool Numerics

35.2.5.1. Defining the Region for the Design Change

To define the region of space where a design change is to be applied, perform the following steps:

1. Click the **Region** tab of the **Design Tool** dialog box.

2. Make a selection from the **Region Geometry** drop-down to specify whether you will define the space as either a **Cartesian** or a **Cylindrical** region type.

3. For either region type, the **Show Bounding Box** option will be enabled by default; this allows you to view the region associated with the settings along with the mesh in the graphics window.

4. For the Cartesian region type, specify the **X Min**, **X Max**, **Y Min**, and **Y Max** (and **Z Min** and **Z Max** in 3D) fields. Alternatively, the following tools are available:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click the **Get Bounds...** button to open the **Bounding Box Definition** dialog box. There you can select the **Zones to be bounded** from the list of zones, and the fields will be populated to create a bounding box sized to the selected zones.
- Click the **Larger Box** and **Smaller Box** buttons to adjust the deformation region size as needed.

Note

Symmetry conditions affect the location and size of the deformation region:

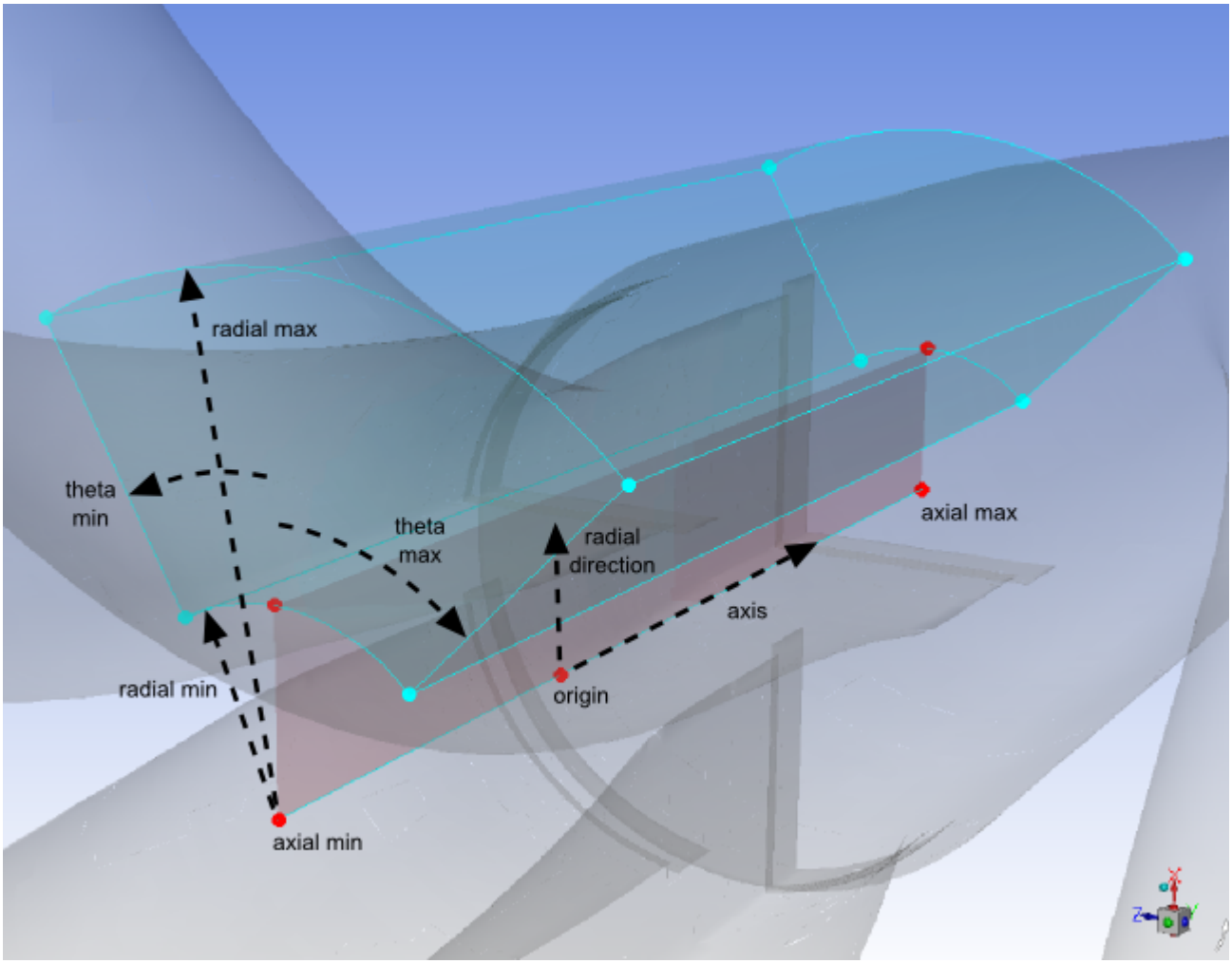
- If you plan to enable the **Symmetry of Motion** options in the **Region Conditions** tab (as described in Defining Region Conditions (p. 2110)), the deformation region should be positioned with forethought given to the effect on the various boundary zones in the region.
- If you have a boundary zone of type **symmetry** in the deformation region, it is recommended that it either is located on a planar boundary of the deformation region, or is centered within the deformation region in a manner that is consistent with the **Symmetry of Motion** option.

---

5. For the cylindrical region type, specify the angular, radial, and (for 3D) axial coordinate ranges; this will require you to define a **Coordinate System**, and then enter the values of the **Region Extent** relative to that system. The fields you will define are illustrated in Figure 35.14: A Cylindrical Region (p. 2110).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 35.14: A Cylindrical Region



Note the following:

- For 2D cases, the axis is defined implicitly as the Z-axis (as determined by the right-hand rule, where in a front view the X-axis is horizontal and the Y-axis is vertical).
- For the angular range (defined by **Theta Min** and **Theta Max**), the positive direction is determined by the right-hand rule about the axis, and the total range must not exceed 360 degrees.
- For 3D cases, the axis and radial direction must not be co-linear.
- Symmetry conditions affect the location and size of the deformation region, as described in the previous step about Cartesian region types. Note that for a cylindrical region type, radial symmetry is not possible.

6. Click **Update Region**. This will set the values for the region boundaries and update the display of the design change region.

35.2.5.2. Defining Region Conditions

1. Click the **Region Conditions** tab in the **Design Tool** dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Specify the control points and motion conditions for each coordinate direction.

   a. Select the number of control **Points**.

      The control points are distributed uniformly within each coordinate range (in the X-, Y-, and Z- directions for a Cartesian region, and the Theta, Radial, and Axial directions for a cylindrical region). The spacing between the control points defines the characteristic spatial scale for the deformation operation. The larger the number of control points, the smaller the spacing and hence, the smaller the spatial scale on which changes can be made. 20 to 40 control points for each coordinate direction is typical.

   b. If you want to prevent deformation of the deformation region in the given coordinate direction, disable **Motion Enabled**.

   c. If you want the motion along a coordinate direction to be uniform in one or more directions, enable the appropriate option (for example, **Invariant in X**).

   d. You can ensure that the deformation is symmetrical relative to the different coordinate directions by enabling options in the **Symmetry of Motion** group box. Symmetry can be imposed in as many directions as desired (though radial symmetry for a cylindrical region is not possible). The symmetry planes will be displayed in the graphics window when you click **Apply**; you should ensure they reflect your intent for the design change, modifying the extents of the deformation region as necessary.

3. The **Region Boundary Continuity** settings control the order of continuity of the deformation process at the boundaries of the deformation region. This is of particular relevance when the boundary of the deformation region intersects a wall that will be deformed. The **Continuity Order** controls the smoothness of the transition from the undeformed wall lying outside the deformation region to the deformed wall within. A value of 1 ensures first-order continuity at the transition, that is, a continuous first derivative of the deformation operation. A value of 2 ensures second-order continuity, and so on.

   By default, the continuity **Definition** is **Uniform** and is enforced at all deformation region boundaries. Alternatively, you can choose **By Boundary** to use different continuity conditions at each boundary. For each boundary (**X-Min**, **X-Max**, and so on) you can specify what order continuity to enforce for each motion direction. You can also specify **In-Plane Motion Only** on that boundary; note that this option may be useful if a deformation region boundary coincides with a boundary zone that is of type **symmetry**.

   The continuity settings will not affect the transition between fixed and movable boundaries within the deformation region, only mesh nodes in the transition zone from inside to outside the deformation region.

4. Click **Apply**.

35.2.5.3. Exporting Sensitivity Data

In order to perform multi-objective optimization, you need to export the sensitivity data for each observable/flow-condition combination that will take part in the optimization.

1. Define the observables of interest as described in Defining Observables (p. 2084).

2. For each observable solve for and export the sensitivity data.

   a. Solve the adjoint as described in Solving the Adjoint (p. 2090).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. In the **Region** tab of the **Design Tool** dialog box, specify the deformation region and click **Export Sensitivities...**.

The exported file contains the sensitivities of the observable for the mesh nodes within the specified deformation region. The deformation regions for the various observables can, in principle, be different. However, it is advisable that there is as much overlap as possible for observables that will be used together in a multi-objective design process.

35.2.5.4. Defining Observable Objectives

The Design Tool computes the optimal mesh deformation to satisfy multiple simultaneous design goals. Among these can be goals for multiple observables, possibly at multiple flow conditions. Once you have obtained the adjoint solutions and displacement sensitivities for the various observables of interest, you can import the data and specify the design objectives for each.

1. Click the **Objectives** tab in the **Design Tool** dialog box.

2. To import previously saved sensitivity data, click **Manage Data...**. In the **Manage Sensitivity Data** dialog box click **Import Sensitivities...** to load previously saved sensitivity files and close the **Manage Sensitivity Data** dialog box.

3. If you want to include computed sensitivity data in the current session that you have not exported yet enable **Include current data**.

4. For each observable objective you can specify the following.

   Objective
   determines how Fluent will attempt to change the value of the observable through the design change. You can choose to increase or decrease the value of the observable optimally, or you can specify a target change in value. You can also select **None** in which case the observable will not be considered in computing the optimal design change, but the predicted change in observable value will still be reported.

   Target/Reference Change
   When the chosen **Objective** is **Target Change in Value**, this field corresponds to the desired change in the value. When the chosen **Objective** is **Maximize** or **Minimize**, the magnitude of this field is a reference weighting for the observable. This can be used to normalize the objectives in cases where you have multiple observables of vastly differing magnitudes, thereby allowing the values entered for **Weight** in the **Design Change** tab to all be of similar scale. You can check **As Percentage** to specify the **Target/Reference Change** as a percentage of the value of the observable.

   Note

   You must click **Apply** for each objective you define before moving on to the next one.

35.2.5.5. Defining Conditions for the Deformation

In the **Design Conditions** tab, you can specify conditions on the mesh morphing such as fixing portions of wall surfaces, prescribing the deformation of surfaces, or setting up bounding planes / surfaces. To create a condition, click **Create...**, select the type of condition in the **Create New Condition** dialog box (you can choose between **fixed-walls**, **bounded-by-plane**, **bounded-by-surfaces**, **prescribed-profile**,

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

rotation, **translation**, and **scaling**), and provide a unique **Name**. You can then define the condition by selecting it in the list on the left, defining the settings (as described in the steps that follow), and clicking **Apply**. If necessary, you can **Rename** a selected condition using the **Rename Constraint** dialog box that opens. Once created, the design conditions can be individually included or excluded from the design change in the **Design Change** tab.

Important

Note that a given surface cannot have multiple conditions of type **prescribed-profile**, **rotation**, **translation**, or **scaling**.

Fixed Walls

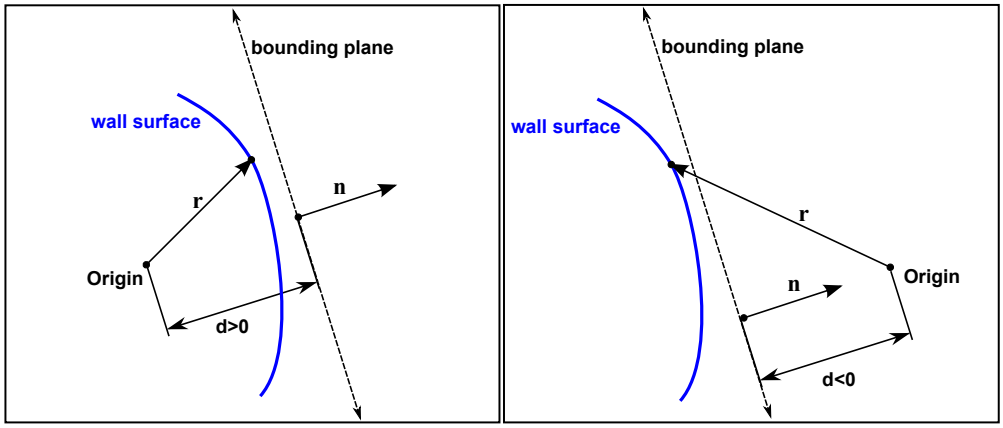
The **fixed-walls** constraint can be applied to clip-surfaces of walls in the deformation region. This offers a fine-grained control over parts of the geometry that should not move. Whereas deselecting a zone in the **Zones To Be Modified** list on the **Design Change** tab will fix an entire wall zone, selecting a clip-surface from in the **Fixed Walls Condition** list will fix only the portion of the wall that forms the clip-surface.

Any clip surfaces you have defined will appear in the **Clip Surfaces** list. You can assign clip surfaces to a fixed walls constraint by selecting them in the list and clicking **Apply**.

Bounding Planes

The **bounded-by-plane** condition allows you to define one or more bounding planes that constrain the geometry deformation in such a way that the geometry cannot cross the given planes, thereby respecting packaging or form-factor constraints. The bounding plane is characterized by a vector normal to the plane and a distance measured from the global coordinate system origin to the plane, along the normal direction. The motion of the selected wall surface(s) will be constrained such that all points on the surface(s) lie on the side of the bounding plane that is opposite to the normal direction.

Figure 35.15: Specifying a Bounding Plane for Design Changes



In Figure 35.15: Specifying a Bounding Plane for Design Changes (p. 2113) the bounding plane normal vector is denoted by $\mathbf{n}$ and the distance is denoted by d. $\mathbf{r}$ is the position vector of a point on the wall surface subject to the bounded-by-plane condition. Note that a positive value for the distance yields a bounding plane that is oriented with its normal pointing away from the origin, while a negative value for the distance yields a bounding plane oriented with its normal direction pointing towards the origin. The constraint equation can be expressed as:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\mathbf{r} \cdot \frac{\mathbf{n}}{|\mathbf{n}|} \leq d$$

1. Select the surfaces (or clip-surfaces) that are subject to the bounding plane condition in the **Bounded Surfaces** list.
2. Enter the components of the bounding plane normal direction.
3. Enter the distance of the plane from the origin (along the normal direction).
4. Click **Apply**.

Bounding Surfaces

The **bounded-by-surfaces** condition allows you to import one or more bounding surfaces that constrain the geometry deformation in such a way that the geometry cannot cross the nearest facet of the bounding surfaces, thereby respecting packaging or form-factor constraints. The bounding surfaces can be read from an `.stl`, `.msh`, or `.cas` file. The deformation of the selected bounded surfaces will be constrained such that all points on the surfaces lie on the sides of the bounding surfaces that have a positive orientation.

1. Select the surfaces (or clip-surfaces) that are subject to the bounded-by-surface condition in the **Bounded Surfaces** list.
2. Use the **Read...** button to import a `.stl`, `.msh`, or `.cas` file that contains the appropriate bounding surface(s). Note that the file you read must have the same dimensionality (2D or 3D) as the working case file.
3. Select the **Imported Surfaces** that will constrain the bounded surfaces.
4. To visually confirm that the selected **Bounded Surfaces** and **Imported Surfaces** are appropriate, click the **Display** button. The imported surfaces will be colored green by default.

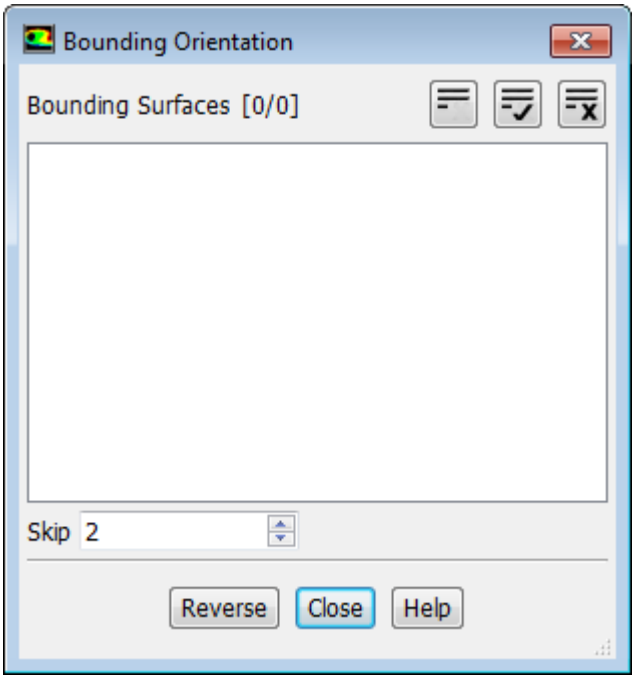
   Note

   When examining the surfaces, it may be helpful to revise display properties of particular surfaces (for example, the color, the level of transparency, and the visibility of the mesh edges or faces). This can be done using the **Scene Description** dialog box, as described in Advanced Scene Composition (p. 1815). Note that you can even revise the display properties of the imported surfaces, the names of which will be displayed with the prefix **bounding-**.

5. To ensure that the **Bounded Surfaces** are constrained on the correct side of the **Imported Surfaces** (that is, the side that has a positive orientation), click the **Orientation...** button. The selected **Imported Surfaces** will be displayed, with their orientation indicated by arrows. In the dialog box that opens (Figure 35.16: The Bounding Orientation Dialog Box (p. 2115)), you can select individual **Bounding Surfaces** and **Reverse** their orientation as necessary; you can also increase the **Skip** value to display less orientation arrows, for cases where the arrow density obscures the orientation. Note that the orientation you define for a particular bounding surface will be used globally for all **bounded-by-surfaces** conditions.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 35.16: The Bounding Orientation Dialog Box



When reviewing the orientation of a bounding surface, you should verify that the facets have a uniform orientation, that is, all of the arrows are on the same side of the surface. Bounding surfaces with facets that have a mixed orientation can affect the convergence and/or accuracy of the calculation.

6. Click **Apply** in the **Design Conditions** tab of the **Design Tool** dialog box to save your settings.

7. If at any point you would like to delete any of the **Imported Surfaces**, you can do so using the Surface Meshes Dialog Box (p. 2807).

Note

When using the **bounded-by-surfaces** condition, it is recommended that you apply preconditioning on the advancement of the process for updating the freeform displacements (by changing the **Preconditioning** field to a positive value in the **Numerics** tab of the **Design Tool** dialog box, as described in Design Tool Numerics (p. 2120)), in order to encourage the stability of the solution. A value on the order of 1 or 10 is recommended.

Prescribed Profiles

You can use the **prescribed-profile** condition to assign a motion profile to a wall zone or zones using a `DEFINE_GRID_MOTION` UDF.

1. Select the wall zones for which to prescribe the displacement in the **Wall Zones** list.

2. Compile your DEFINE_GRID_MOTION UDF in Fluent. (Refer to `DEFINE_GRID_MOTION` in the *Fluent Customization Manual*).

3. Select your UDF in the **Deformation profile** drop-down list.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Specify the **Scale Factor** to be applied to the deformation profile. Alternatively, you can specify or create an input parameter to use for the **Scale Factor** by clicking the **p** button next to the **Scale Factor** field.

5. Click **Apply**.

Rotation

The **rotation** condition allows you to specify that surfaces undergo rotation.

1. Select the surfaces (or clip-surfaces) you want to rotate in the **Surfaces** list.

2. Enter the rotation **Angle** in degrees. Note that you can enter a set value, or use the P button to assign an input parameter (for example, as part of a parametric study managed by Workbench); see Select Input Parameter Dialog Box (p. 2384) for details.

3. Define the **Origin** about which you want the surfaces to rotate.

4. For 3D cases, define the **Axis** of rotation. For 2D cases, this is automatically defined as the positive Z axis.

5. Click **Apply**.

Translation

The **translation** condition allows you to specify that surfaces undergo translation.

1. Select the surfaces (or clip-surfaces) you want to translate in the **Surfaces** list.

2. Define the **Displacement** to be applied along each coordinate axis. Note that for each field you can enter a set value, or use the P button to assign an input parameter (for example, as part of a parametric study managed by Workbench); see Select Input Parameter Dialog Box (p. 2384) for details.

3. Click **Apply**.

Scaling

The **scaling** condition allows you to specify that surfaces are scaled along one or more axes.

1. Select the surfaces (or clip-surfaces) you want to scale in the **Surfaces** list.

2. Select the scaling **Type** and define the associated settings:

   - **Radial**

     This type scales the surfaces radially about the **Origin**, using a single scaling **Factor** along each of the coordinate axes. The size of the surfaces will change, though not the shape.

   - **In-Plane**

     This type is available for 3D cases, and scales the surfaces about the **Origin** using a single scaling **Factor** along the two axes of a plane (as defined by the **Normal Direction**).

   - **Axial**

     This type scales the surfaces about the **Origin** using a scaling **Factor** along a single **Axis**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **General**

  This type scales the surfaces about the **Origin** using a separate scaling **Factor** for each of the defined axes. For 2D cases, you define one **Axis**, and the second axis is automatically defined as being perpendicular; for 3D cases, you define **Axis 1** and **Axis 2** (which should not be parallel), and the third axis is defined automatically as the cross product. Note that the surfaces will be scaled along each axis sequentially, so the order will be relevant for non-orthogonal axes.

For all types, the scaling **Factor** must be positive. A value of 1 results in no change, while a value greater or less than 1 results in stretching or shrinking, respectively. Note that you can enter a set value, or use the P button to assign an input parameter (for example, as part of a parametric study managed by Workbench); see Select Input Parameter Dialog Box (p. 2384) for details.

3. Click **Apply**.

35.2.5.6. Shape Modification

The procedure for computing the optimal design change and deforming the mesh if as follows:

1. Click the **Design Change** tab.

2. Select those boundaries that are free to be partially constrained or deformed in the **Zones To Be Modified** field. Boundaries that are not selected in this box that intersect the control volume will be constrained not to move. This is useful if there are walls that, for design reasons, must remain fixed and not move.

3. Select the constraining and/or deformation conditions (Defining Conditions for the Deformation (p. 2112)) to be applied to the design change in the list of **Applied Conditions**. Note that a given surface cannot have multiple conditions of type **prescribed-profile**, **rotation**, **translation**, or **scaling**.

4. For each observable, specify its **Weight** in the optimization calculation. The weights are applied to the **Target/Reference Change** you specified in the **Objectives** tab. That is, if you have two observables that are each given a **Weight** equal to 1, Fluent will attempt to find an optimal design change that alters the observable values in proportion to their respective values for **Target/Reference Change.**

   Note

   Weights are only required if there are two or more objectives that do not have explicit target changes specified.

5. Specify the deformation parameters.

   a. Choose a **Freeform Scaling Scheme** and specify the **Freeform Scaling Factor**. This setting controls how the overall scaling factor for the design change magnitude is interpreted.

      Control-point spacing
      The magnitude of the freeform deformation will be based on the control-point spacing. That is, a **Freeform Scale Factor** of `1.0` leads a to maximum control point movement on the order of one sub-cell of the control volume. The sub-cell size is determined by a uniform sub-division of the morphed region based on the number of control points specified (Defining Region Condi-

tions (p. 2110)). A value of `1.0` is a reasonable default to preserve mesh quality during the morphing process.

Objective reference change
The magnitude of the freeform deformation will be based on the **Target/Reference Change** value in the **Objectives** tab. That is, for a single observable optimization with **Freeform Scale Factor** of `1.0`, the design change computed will yield an expected change in the observable equal to the **Target/Reference Change** for the observable.

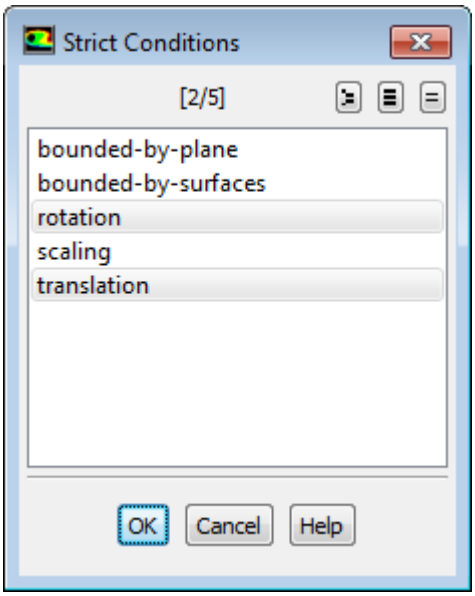
b. If you have disabled **Auto-Select Smoothness** in the **Numerics** tab (Design Tool Numerics (p. 2120)) you can also specify a value for **Smoothness** of the deformed geometry. A lower value will allow locally sharper changes in the geometry, potentially at the expense of manufacturability.

c. If you used an input parameter as part of the setup of a deformation condition in the **Design Conditions** tab, the initial value will be displayed and is available for modification.

6. Click the **Check** button to verify that your constraining and/or deformation conditions are set up in such a way as to produce good results. A report will be printed in the console, summarizing the number of control points defined in the **Region Conditions** tab that are active (that is, not disqualified by other constraints) and listing conflicts between multiple constraints / deformations applied on a single node (including the **Design Conditions** and the continuity constraint applied along the boundary of the morphing region). The conflicts are categorized as being either of type `Fatal` (which indicates the solution is unlikely to converge or be desirable) or `Possible` (which may be allowable, depending on the details of the applied conditions).

7. By default, the condition or conditions specified for a given zone (including the selected **Applied Conditions**, as well as the fixed condition applied to the unselected **Zones To Be Modified**) are not applied to every node, but instead to a subset of nodes distributed throughout the zone; this is to avoid overly constraining the zone. Consequently, some nodes will not strictly adhere to the condition, even if you have defined the constraint tolerance to be small or the residuals are small. To specify that all nodes of the zone strictly obey the condition, click the **Strict Conditions...** button and use the dialog box that opens (Figure 35.17: The Strict Conditions Dialog Box (p. 2118)).

Figure 35.17: The Strict Conditions Dialog Box



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select the conditions for which you want strict enforcement and click **OK**. For unselected conditions, only the default subset of nodes will strictly obey the condition. Note that when you apply strict enforcement to a case in which the **Objective** is **Target Change in Value** (as described in Defining Observable Objectives (p. 2112)), the desired change in the value may not be achieved.

Note

Strict enforcement is a direct modification of the surface mesh nodes to satisfy the design condition, and no modification is applied on the interior mesh. Consequently, when the boundary node modification is large, it may deteriorate the mesh or even cause negative volumes. Therefore you should use this condition carefully: the design condition should reach a good convergence and there should be enough control points.

When you click the **Modify Mesh** button (as described in a later step), the maximum offset for each condition (that is, the maximum distance a node has deviated from the prescribed condition) will be printed to the console.

8. Click **Calculate Design Change** to compute the optimal design change. The **Expected change** for each observable will be reported in the **Design Tool** dialog box.

9. (optional) Click **Write Expected Change...** to write to file the adjustment in the observable that is expected due to the change defined by the current settings.

10. (optional) Click **Export Displacements...** to write to file the optimal surface displacement field (overwriting any pre-existing file of the same name). An example of the format of the text file that is written (for three-dimensions) is as follows:

```
Line 1: n
Lines 2 to n+1: x y z dx dy dz
```

11. The planned change to the geometry can be visualized by plotting the optimal displacement vector field or a contour plot of surface deformation.

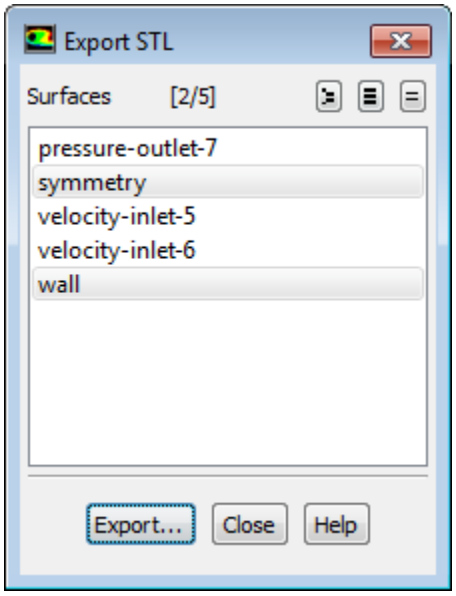
12. Click **Modify Mesh** to deform the boundary and interior mesh with the optimal design change. If you determine that the change is unsatisfactory, you can click **Revert** to undo the mesh changes.

Upon successful modification of the geometry of the system, the modified shape can be viewed using the usual commands for viewing the mesh in ANSYS Fluent. After the mesh has been modified according to the control settings, the original flow calculation can be restarted and converged with the modified geometry. Upon successful convergence, the observable value should now have changed by an amount similar to that reported when the **Expected Change** button was clicked prior to deforming the geometry.

For 3D cases, you have the option of exporting the surfaces from your modified geometry as an `.stl` file, for use in other software packages. Click the **Export STL...** button near the bottom of the **Design Change** tab, select the **Surfaces** of interest in the dialog box that opens (Figure 35.18: The Export STL Dialog Box (p. 2120)) and **Export...** them.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 35.18: The Export STL Dialog Box



35.2.5.7. Design Tool Numerics

Smoothness

As described in Smoothing and Mesh Morphing (p. 2080), the mesh morphing is controlled by two superposed smoothing bases: the Bernstein polynomial basis which control the large scale smooth deformation and the B-spline basis which controls fine-scale deformation. The choice of smoothness determines the relative weighting of these bases and therefore the spatial scale of the design change. A large value of smoothness biases the design change to one in which optimal displacements vary in space in a manner dominated by the Bernstein polynomial basis. That is, they have a high degree of smoothness. A smaller value of smoothness leads to a recommended design change with smaller spatial scales.

The ability to vary the smoothness parameter is particularly important when prescribed displacements and design conditions are applied. Deformation at smaller spatial scales relieves the stiffness that would otherwise cause numerical challenges as well as lead to poor quality of the computed design and mesh.

By default, Fluent will automatically choose the smoothness. In the Numerics tab, you can disable **Auto-Select Smoothness** in which case a **Smoothness** input will be made available on the **Design Change** tab.

Calculation Numerics

Note

If no **Design Conditions** are imposed, and if there are no walls that are treated as fixed which intersect the region being modified, the numerical settings described below are not used.

In many cases, no adjustment of the detailed numerics will be required for the design tool to compute an optimal solution. However, if the problem under investigation involves many Design Conditions and/or large displacements, the settings may need to be adjusted.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Prescribed Motions

Max. Iterations
The maximum number of inner iterations that will be performed in an effort to compute displacements from prescribed changes.

Constraint Relaxation
The under-relaxation factor that is applied to the field to converge the prescribed displacements. This takes a value between 0 and 1.

Preconditioning
A value of 0 yields the most aggressive advancement, while a larger value encourages more stable, albeit slower, advancement of the process for updating the prescribed displacements. A value of order 1 is typical in the event that preconditioning needs to be applied.

Freeform Motions

Max. Iterations
The maximum number of inner iterations that will be performed in an effort to compute freeform displacements that satisfy the applied **Design Conditions**.

Constraint Relaxation
The under-relaxation factor that is applied to the field to converge the constrained freeform displacements. This takes a value between 0 and 1.

Preconditioning
A value of 0 means that no preconditioning is applied, while a larger value encourages more stable, albeit slower, advancement of the process for updating the freeform displacements. A value of order 1 is typical in the event that preconditioning needs to be applied.

Parameter Relaxation
For design conditions such as the Bounded-By-Plane condition there are additional parameters that are implicit in the calculation. The values of these parameters are computed as part of the solution process. The value of the parameter relaxation defines the rate at which these parameters are corrected as the calculation progresses. This takes a value between 0 and 1.

Tolerances

Constraints
The tolerance for convergence of the residuals associated with the design conditions themselves, or fixed zone conditions.

Parameters
The tolerance for convergence of the parameters associated with the design conditions.

Using a value of 1 for **Constraint Relaxation** and a value of 0 for **Preconditiong** yields the most aggressive behavior. This can result in the fewest number of iterations to converge a constrained problem, at the expense of some added numerical effort and memory in preparing the preconditioning. For 2D and small 3D problems the added cost in time and memory is typically small. For larger problems involving multiple constraints a non-zero **Preconditioning** and **Constraint Relaxation** less than 1 may be preferable. The execution time to set up the preconditioning will be less, as will the memory usage. Calculation stability can also be improved. The number of iterations required to converge the problem will increase. However, this can prove to be an effective strategy that results in an overall shorter time needed to compute the design change.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.3. The Mesh Morpher/Optimizer

The mesh morpher/optimizer offers a variety of non-gradient-based optimization algorithms to optimize the geometric shape of a system [47] (p. 2889). The optimization can be performed for a wide variety of physics and quantities of interest to be optimized.

For additional information, see the following sections:

35.3.1. Limitations
35.3.2. The Optimization Process
35.3.3. Optimizers

35.3.1. Limitations

Note the following limitations of the ANSYS Fluent mesh morpher/optimizer:

- The mesh morpher/optimizer should not be used with dynamic or sliding mesh problems.
- Any geometrically evaluated parameters required for flow iterations (for example, view factors for the surface to surface (S2S) radiation model) will need to be recomputed at the very least when the optimization is complete. If the objective function is affected by one of these geometrically evaluated parameters, then such parameters will need to be recomputed after each design change; for example, the view factors need to be recomputed after each design change when the objective function is a function of temperature.
- Arbitrarily shaped deformation regions are not supported.

35.3.2. The Optimization Process

All optimization problems require that you identify parameters that can be modified in order to reach the optimized solution. In the case of the mesh morpher/optimizer, it is the geometry that must be parameterized. Geometric parameterization for general shapes used in CFD can be very complicated, due to the large variety of shapes available in engineering applications. In order to minimize such complications in your ANSYS Fluent simulation, the problem of shape parameterization is reduced to a problem of the parameterization of changes in the geometry.

The next essential requirement for mesh morphing is a tool that can smoothly alter the shape, irrespective of the underlying mesh topology. In ANSYS Fluent, designated deformation regions are manipulated via displacements applied to a set of control points. The mesh region that is to be deformed is defined by a "box" (that is, a rectangle for 2D cases and a rectangular hexahedron for 3D), and the control points must be located within the box. The displacements of the control points are the result of user-defined motions (each of which involves a parameter value and other directional settings) and these displacements are then applied to the mesh as a smooth deformation by either interpolating the displacement based on radial basis functions or using the tensor product of Bernstein polynomials.

The next requirement is to have an optimizer that is robust enough to handle a wide range of problems. By coupling such optimizers with your CFD analysis, you can greatly improve your design with minimal intervention. You can use built-in optimizers to vary the parameter values along the prescribed directions and within defined bounds, in order to satisfy a condition specified by an objective function. The mesh morpher/optimizer provides you with access to six optimizers that are not based on gradients. Otherwise, you can manually specify the deformation (that is, define both the parameter values and the directions) and analyze the results; you also have the option of using Design Exploration in ANSYS Workbench to easily explore the impact of a variety of parameter values.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.3.3. Optimizers

The built-in optimizers used as part of the mesh morpher/optimizer capability in ANSYS Fluent use direct search methods for optimization. Direct search methods are zeroth order, as they only use the objective function values for optimization. The direct search methods do not use the derivatives.

The following is a list of advantages of direct search methods:

- Direct search methods do not require derivatives for optimization.
- Direct search methods are robust for problems with discontinuities and in situations where the derivative computation is not possible or unreliable.

The following is a list of disadvantages of direct search methods:

- Convergence proof is not clearly defined.
- The rate of convergence can be very slow.

General explanations of the six different built-in optimizers are provided in the following sections:

35.3.3.1. The Compass Optimizer
35.3.3.2. The NEWUOA Optimizer
35.3.3.3. The Simplex Optimizer
35.3.3.4. The Torczon Optimizer
35.3.3.5. The Powell Optimizer
35.3.3.6. The Rosenbrock Optimizer

35.3.3.1. The Compass Optimizer

In the Compass optimizer [53] (p. 2889), the parameters are adjusted one by one until the objective function is minimized. This optimizer starts with a given value and then evaluates the function value in all the basic directions. The direction here refers to the positive and negative increments to the initial parameter values. If there is a reduction in the function value, then that point becomes an improved point. If there is no improvement in the function values, then the step length is reduced by half and the search is repeated in all directions. The algorithm terminates when the step size falls below a certain tolerance.

The Compass optimizer initially makes rapid progress towards the solution. While this method might quickly approach the minimum value of an objective function, it may be slow to detect this fact. It may also converge very slowly if the level sets of the objective function are extremely elongated.

35.3.3.2. The NEWUOA Optimizer

The NEW Unconstrained Optimization Algorithm (NEWUOA) optimizer [89] (p. 2891) attempts to find the least value of a function $F(x)$, where x is a parameter vector of n dimensions. At the start of every iterative step, a quadratic model approximation ($Q \sim F$) is constructed and the minimization is performed on a trust region (that is, a region around the parameter that is limited by an initial parameter variation). The perturbation to the parameter value needed to obtain the lowest value of Q is evaluated during the iteration, and correspondingly, the new least value of $F(x)$ is obtained. The iterative process continues until the trust region is reduced to the optimizer convergence criterion, and the optimizer exits with the least value of the objective function. This optimization algorithm can be applied to any setup case of the mesh morpher/optimizer.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The NEWUOA optimizer provides the least value of the objective function in a manner that is highly accurate and robust. The main advantage of this optimizer is that it is very fast; hence, this optimizer is recommended for problems that have a large number of parameters.

35.3.3.3. The Simplex Optimizer

The Simplex optimizer is also referred to as the Downhill Simplex optimizer [68] (p. 2890), [47] (p. 2889) and the Nelder-Mead method [74] (p. 2891). It is based on the idea of geometric simplexes; for example, a 2D simplex is a triangle, and a 3D simplex is a tetrahedron.

For optimization purposes, ANSYS Fluent requires that simplexes are regular polyhedra (that is, not degenerate polyhedra with collapsed sides). Each vertex of the geometric simplex represents one function evaluation (which in this case is one CFD run), and the number of vertices corresponds to the number of parameters. For the free-form deformation method that is used by ANSYS Fluent to parameterize changes in shapes, the number of active control points will determine the number of vertices of the geometric simplex.

Minimization of the objective function is performed based on a set of the rules about the "quality" of each vertex. The vertex quality is the value of the function evaluated for each position of the control point. A set of geometric operations such as reflection, expansion, contraction, and shrinking are performed in order to find the region in which to look for the minimum of the function. Because optimization here is formulated as a minimization problem, the simplex optimizer algorithm seeks the "worst" vertex, that is, the vertex that has the largest value when the corresponding parameter is evaluated. By performing the reflection around the center of the gravity, the new value of the function is obtained after performing the CFD run. Similarly, the operations of expansion, contraction, and shrinking are used to obtain the minimum of the function.

The Simplex optimizer is known to work well, but it suffers from the large number of function evaluations. It also requires smooth objective functions for convergence.

35.3.3.4. The Torczon Optimizer

The Torczon optimizer [125] (p. 2893) is a slightly modified version of the simplex optimizer described previously. Given an initial vertex, this optimizer tries to find a better vertex that has a function value that is strictly less than the function value at the previous best vertex. There are three possible trial steps: the rotation step, the expansion step, and the contraction step. The algorithm always computes the rotation step and then tests to see if a new best vertex has been identified. If it has, then the expansion step is computed. Otherwise, the algorithm computes and automatically accepts the contraction step.

35.3.3.5. The Powell Optimizer

For the Powell optimizer, the method used is based on the number of dimensions of the problem. For optimization problems of single dimension (that is, problems with a single parameter), the golden section search algorithm [90] (p. 2891) is used to find the optimal value for the objective function. In this algorithm, the optimal value is found by reducing the size of the bracketing triplet, until the size of the bracket (that is, the distance between the outer points of the triplet) reaches a certain tolerance level. In all the cases, the middle point of the new triplet is determined to be the best value obtained so far.

For multidimensional problems (that is, problems with multiple parameters), the minimum value and the largest decrease is found using the golden search algorithm for each dimension, in order to find the conjugate directions. A conjugate direction is a direction that when searched will not alter the minimum value attained by the previous movement in another direction—that is, a direction in which

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the gradient is perpendicular to the first direction. After finding the N linearly independent, mutually conjugate directions, one pass will find the exact minimum value. For functions that are not quadratic, repeated cycles of N line minimizations will converge to minimum.

35.3.3.6. The Rosenbrock Optimizer

In the Rosenbrock optimizer [96] (p. 2892), after the initial direction is found, multiple steps are taken in that direction until the least value is attained. The process starts with an arbitrary length e. If this initial step succeeds (that is, the new value of the function is less than or equal to the old value), the length e is multiplied by α, where α is more than 1. If the steps fails, the length e is multiplied by β, where β is between -1 and 0. The direction or the parameter that must be modified is determined by advancing all the parameters by the step length e and then selecting the best among those that yield a function value that is less than the previous value. After that point is accepted as the best point, the process is repeated. These steps continue to repeat until e becomes so small so that any further change in the value of e does not significantly reduce the value of the function.

35.4. Using the Mesh Morpher/Optimizer

The procedure for setting up and using the mesh morpher/optimizer for shape optimization is as follows:

1. Read the case into ANSYS Fluent.

   File → **Read** → **Case...**

2. If you want to use one of the built-in optimizers rather than specifying the deformation manually or using Design Exploration in ANSYS Workbench to explore multiple deformation scenarios, you will need to provide an objective function in one of three ways: either as a user-defined function (UDF), a Scheme source file, or a customized function that is based on output parameters (that is, values from flux, force, surface integral, or volume integral reports). The goal of the optimizer is to deform the mesh in such a way that this objective function is minimized. ANSYS Fluent will run the solution for a given design stage until convergence is reached, and then check to see if the objective function is satisfied. If the objective function is not satisfied, ANSYS Fluent will proceed to the next design, and so on, until convergence is achieved from the point of view of the optimizer. You can specify the optimizer convergence criteria in step 12.d.iii.

   Note that if you plan to use the **newuoa** optimizer, you must define the objective function as a UDF.

   a. If you want to provide the objective function as a user-defined function (UDF), perform the steps that follow. For more information about UDFs, see the separate Fluent Customization Manual.

      i. Write a UDF using the `DEFINE_ON_DEMAND` macro to define the objective function. The function name must be `objective_function`. At the end of the objective function, the rpvar `morpher/objective_function` must be set to the `(current - target)` value. It is this value that the optimizer will attempt to minimize.

      ii. Compile the UDF using the **Compiled UDFs** dialog box. Make sure `libudf` appears as the **Library Name**.

      User Defined → **User Defined** → **Functions** → **Compiled...**

   b. If you want to provide the objective function as a Scheme source file, perform the following steps:

i. Write a Scheme source file to define the objective function. The procedure name must be `objective-function`. At the end of the objective function, the rpvar `morpher/objective-function` must be set to the `(current - target)` value. It is this value that the optimizer will attempt to minimize.

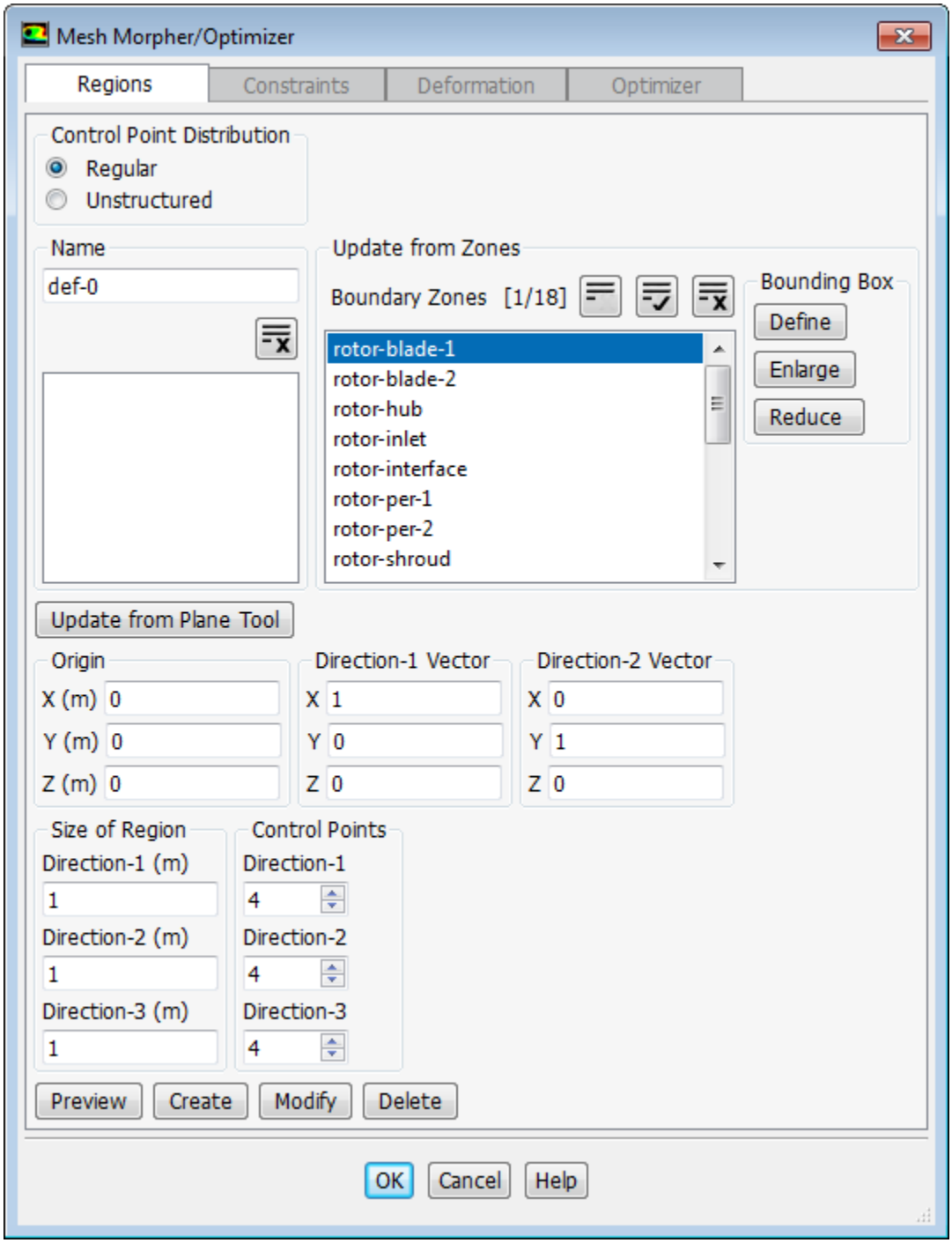
ii. Load the Scheme source file (see Reading Scheme Source Files (p. 57) for details).

3. Open the **Mesh Morpher/Optimizer** dialog box by clicking **Optimizer...** in the **Design** ribbon tab (**Parameter-Based** group box).

   The first time you click **Optimizer...** in a session, a question dialog box will ask you whether you want to enable the mesh morpher/optimizer. Click **Yes** to load the libraries and open the **Mesh Morpher/Optimizer** dialog box (Figure 35.19: The Regions Tab of the Mesh Morpher/Optimizer Dialog Box (p. 2127)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 35.19: The Regions Tab of the Mesh Morpher/Optimizer Dialog Box



4. Specify the manner in which you would like to define the locations of the control points, by making a selection from the **Control Point Distribution** list in the **Regions** tab of the **Mesh Morpher/Optimizer** dialog box (Figure 35.19: The Regions Tab of the Mesh Morpher/Optimizer Dialog Box (p. 2127) and Figure 35.20: The Regions Tab of the Mesh Morpher/Optimizer Dialog Box for an Unstructured Distribution (p. 2129)). Your selection here will also determine the kinds of motions you can apply to the control points (in the **Deformation** tab). You have the following choices:

   - **Regular**

For this option, the control points are spread in a regular distribution throughout the entire deformation region; you only have to define the number of control points along each of the defining direction vectors. Note that as part of this option, the motion of groups of control points will have to be defined as a translation, as the ability to define rotational or radial motions is not available.

Locating control points in this manner is easy and simple, though this comes at the cost of being able to precisely define control points on specific mesh nodes and boundaries. This option can be a good choice in cases where you are able to allow the entire mesh within the region to be deformed—for example, internal flows within a pipe or duct.

Note that as part of this option, the mesh deformation is based on Bernstein polynomials. While such a direct morphing approach is computationally less expensive than using radial basis functions, it does not provide you with precise control of a particular mesh node.

- **Unstructured**

  For this option, you will define the locations of the control point by right-clicking with the mouse on boundary zones, distributing an approximate number throughout a zone, or by entering coordinates. Note that as part of this option, the motion of groups of control points can be defined as a translation, as a rotation about a point / axis, or radially about a point / axis.

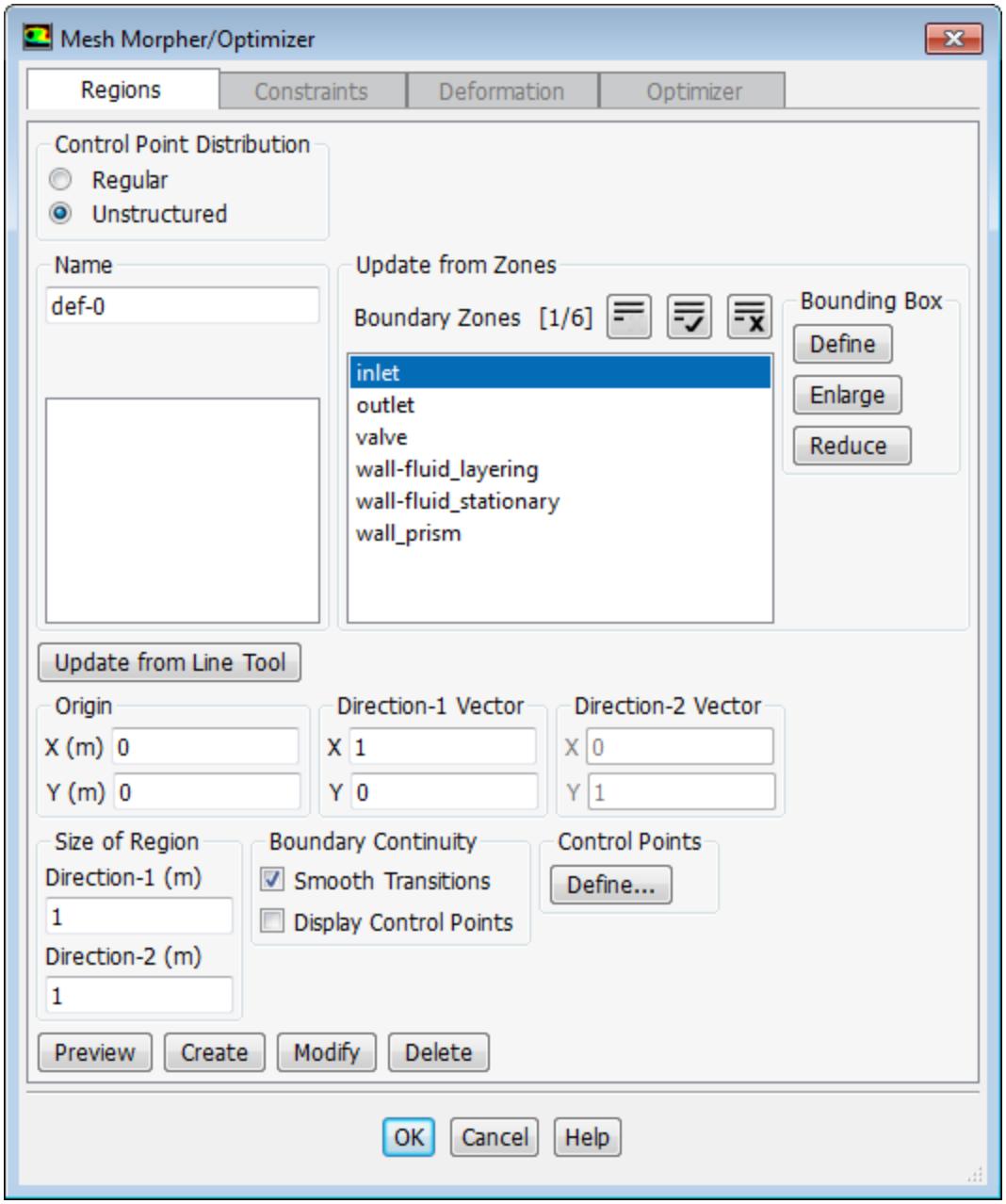
  This option provides great precision with regard to the location of the control points; this allows you to morph particular nodes and regions of the mesh, as well as to preserve features within the deformation region (as you can make nodes stationary by applying a parameter value of zero). This option also allows you to easily define twisting and axisymmetrical motions. This may be a good choice for external flows and aerodynamic applications.

  Note that as part of this option, the mesh deformation is based on radial basis functions. While this is computationally more expensive than using Bernstein polynomials (because it involves the solution of a system of equations), it allows you to have precise control of a particular mesh node if you have created a control point with the same coordinates.

Note that all of the deformation regions that you define in the next step must use the same **Control Point Distribution**; if you change this selection at any point, all of the settings in the **Regions** tab will be deleted.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 35.20: The Regions Tab of the Mesh Morpher/Optimizer Dialog Box for an Unstructured Distribution



5. Define the region(s) of the domain where the mesh will be deformed in order to optimize the shape, by performing the following steps. Each deformation region will be defined as a "box", that is, a rectangle for 2D cases and a rectangular hexahedron for 3D cases.

Important

It is recommended that you define each deformation region to be bigger than the area of interest, in order to maintain proper continuity between deforming and non-deforming regions.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. In the **Regions** tab, enter a name for a deformation region in the text-entry box at the top of the **Name** group box.

b. You have the option of using the boundary zones to create a bounding box for the deformation region; this box can represent the final scope of the region, or it can act as a starting point that is further refined by manually editing the origin, direction vectors, and size of the region (as described in the steps that follow). Perform the following steps in the **Update from Zones** group box:

   i. Select the zones from the **Boundary Zones** list that best represent the extents of the deformation region you would like to create.

   ii. Click the **Define** button in the **Bounding Box** group box to update the values in the **Origin**, **Direction-1 Vector**, **Direction-2 Vector** (for 3D cases), and **Size of Region** group boxes. Note that the bounding box will be previewed in green in the graphics window when you click any of the **Bounding Box** buttons.

   iii. You can increase or decrease the size of the bounding box by clicking the **Enlarge** and **Reduce** buttons, respectively. Note that you have the option of setting the scaling factors associated with these buttons via the following text commands:

   `define` → `mesh-morpher-optimizer` → `region` → `scaling-enlarge`

   `define` → `mesh-morpher-optimizer` → `region` → `scaling-reduce`

c. You have the option of using the line tool (for 2D cases) or the plane tool (for 3D cases) to define the direction vectors of the deformation region; the values defined can represent the final components of the vectors, or they can act as a starting point that is further refined by manually editing the direction vectors (as described in the steps that follow). Set up the line tool or the plane tool (as described in Using the Line Tool (p. 1720) and Using the Plane Tool (p. 1724), respectively), and then click the **Update from Line Tool** or **Update from Plane Tool** button to update the values in the **Direction-1 Vector** and (for 3D cases) **Direction-2 Vector** group boxes.

d. Define an origin for the deformation region by entering the Cartesian coordinates of a point in the **X**, **Y**, and (for 3D) **Z** number-entry boxes in the **Origin** group box. Note that as you enter values, you can view the resulting bounding box (in green) by clicking the **Preview** button.

e. Define the first direction vector of the deformation region relative to the **Origin** coordinates, by entering values in the **X**, **Y**, and (for 3D) **Z** number-entry boxes in the **Direction-1 Vector** group box.

   For 2D cases, ANSYS Fluent will automatically define the second direction vector of the deformation region to be perpendicular to the **Direction-1 Vector**, and will display the components in the uneditable **Direction-2 Vector** group box.

f. For 3D cases, define the second direction vector of the deformation region relative to the **Origin** coordinates, by entering values in the **X**, **Y**, and **Z** number-entry boxes in the **Direction-2 Vector** group box. If the vector you define is not perpendicular to the **Direction-1 Vector**, ANSYS Fluent will automatically redefine the second vector to be the projection of the **Direction-2 Vector** you entered onto a plane that is perpendicular to the **Direction-1 Vector**. Based on this definition, the **Direction-2 Vector** you define cannot be colinear with the **Direction-1 Vector**.

   The third direction vector of the deformation region will be automatically defined as the cross product of the first and second direction vectors.

g. Define the overall dimensions of the deformation region by entering length values for **Direction-1**, **Direction-2**, and (for 3D cases) **Direction-3** in the **Size of Region** group box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

h. Create control points within the deformation region.

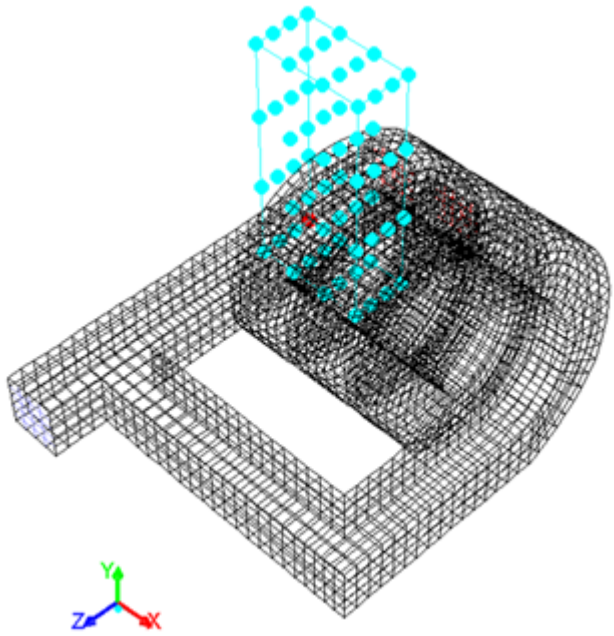
i. If you selected **Regular** for the **Control Point Distribution**, define the number of control points you want along each direction vector of the deformation region by entering values for **Direction-1**, **Direction-2**, and (for 3D cases) **Direction-3** in the **Control Points** group box. The total number of control points for the region will be the product of the numbers you enter. Increasing the number of control points allows you greater control of the deformation, but also increases the computational expense.

Important

When you define multiple deformation regions, you must ensure that all the deformation regions have the same total number of control points.

Save the settings you have created for the deformation region by clicking the **Create** button. The name of the deformation region will be added (and selected) in the selection list at the bottom of the **Name** group box, and the control points and bounding box of the deformation region will be displayed in blue in the graphics window (see Figure 35.21: Displaying the Control Points for a Regular Distribution (p. 2131)). If you do not want the control points and bounding box displayed in the graphics window, you can deselect the item by clicking the button located above the right side of the **Name** selection list.

Figure 35.21: Displaying the Control Points for a Regular Distribution



A new default name will be automatically entered in the text-entry box at the top of the **Name** group box, in preparation for any deformation regions you want to create in the future.

ii. If you selected **Unstructured** for the **Control Point Distribution**, click the **Create** button to save the settings for the deformation region, as you need an existing region to be able to create control points. The name of the deformation region will be added (and selected) in the selection list at the bottom of the **Name** group box, and the bounding box will be displayed in blue in the graphics window (see Figure 35.23: Displaying the Control Points for an Unstructured Distribution (p. 2134)).

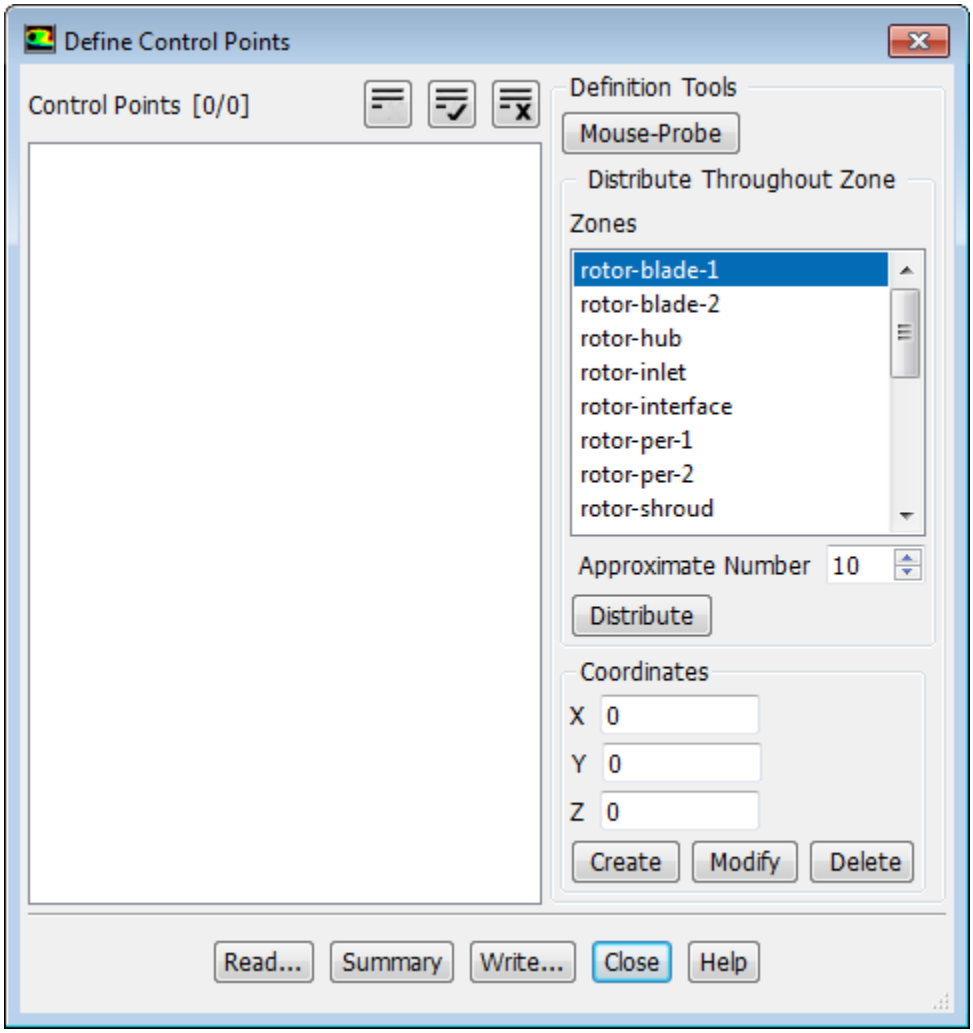
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If at any point you do not want the bounding box displayed in the graphics window, you can deselect the item by clicking the button located above the right side of the **Name** selection list.

Next, examine the settings in the **Boundary Continuity** group box. By default, the mesh morpher/optimizer is set to provide smooth mesh transitions wherever the mesh intersects with the limits of the deformation region (that is, the sides of the bounding box). This is accomplished through the creation of stationary control points at those intersections; by default, these control points are hidden, though you can view them by enabling the **Display Control Points** option. If you would like to allow more abrupt mesh transitions at the boundaries of the region, you can disable the **Smooth Transitions** option.

The next step is to create control points. Click the **Define...** button in the **Control Points** group box to open the **Define Control Points** dialog box (Figure 35.22: The Define Control Points Dialog Box (p. 2132)).

Figure 35.22: The Define Control Points Dialog Box



Use the **Define Control Points** dialog box to create control points within the region. You have the following options:

- Click the **Mouse-Probe** button and then probe (using the right mouse button, by default) within the deformation region. For 3D cases, if you probe on a displayed boundary, the control point

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

will be located at that point on that boundary. Note that it may be helpful to revise what surfaces are displayed using the Mesh Display Dialog Box (p. 2163).

- Make a selection from the **Zones** list, enter an **Approximate Number** of control points that you want on that zone, and click the **Distribute** button. This method uses Binary Space Partitioning (BSP) to agglomerate the cell faces of the zone within the bounding box based on the intended number of control points, and then a control point is assigned to the node that is closest to the centroid of each agglomeration. While this method is convenient when you want a large number of control points or want to morph entire boundary zones or surfaces, there are some limitations to BSP: the number of control points may exceed the value you enter, and the control points will not be as evenly distributed on curved surfaces when compared to flat surfaces.

- Enter **X**, **Y**, and **Z** coordinates and click the **Create** button. This method allows you to locate a control point directly on a mesh node, if you have first obtained the vertex position (which can be displayed in the console if you select **long description** from the **Probe** drop-down list in the **Viewing** tab (**Mouse** group box) and then probe near the node with the mouse).

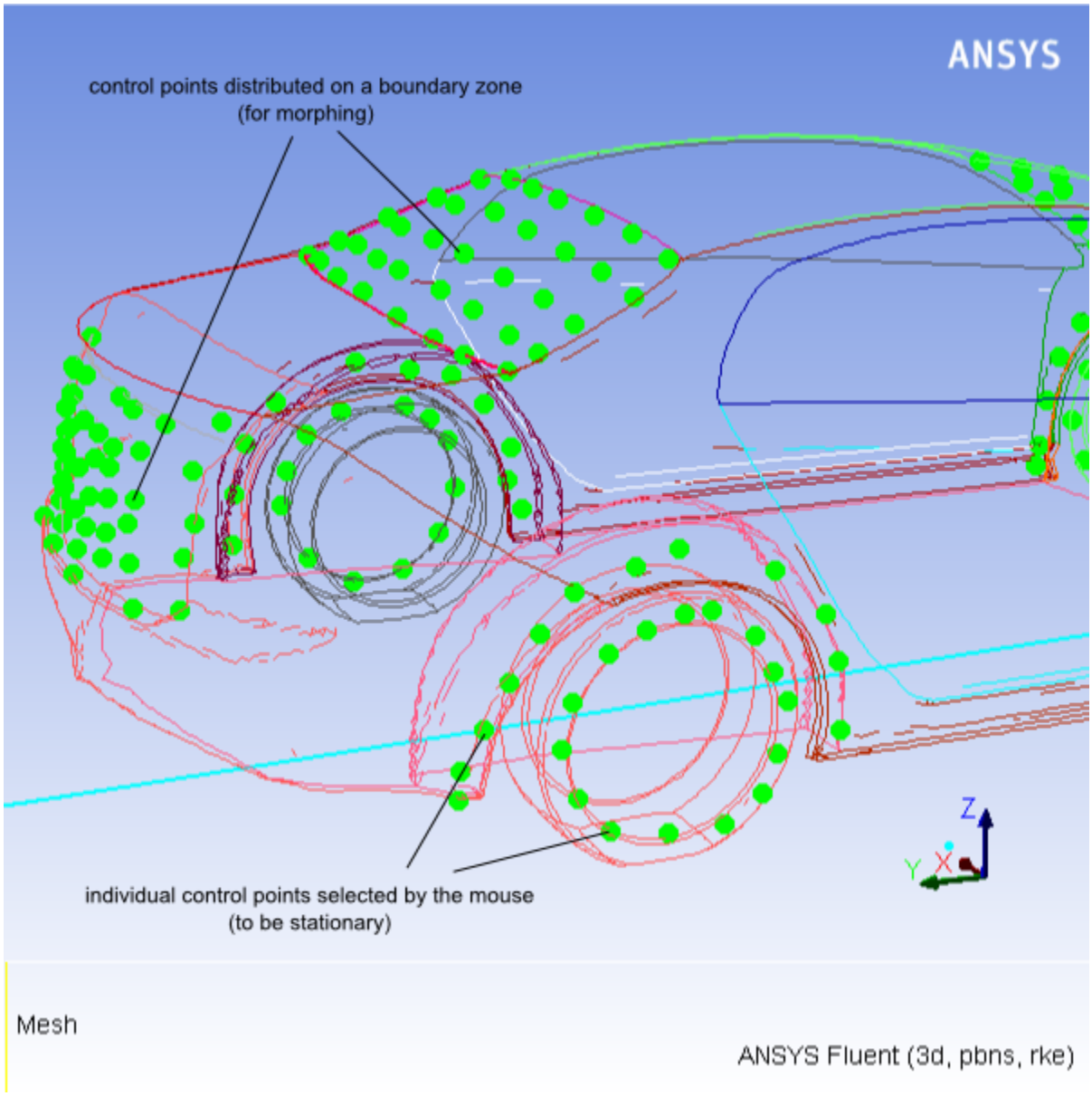
- If you have previously generated an ASCII text file that contains control point definitions, you can read it by clicking **Read...** and using the dialog box that opens. Using a text file can be helpful when you have a large number of points. It is not necessary to create the text file from scratch, as you can create a simplified text file with sample data using the **Write...** button, and then define the bulk of the control point locations using a spreadsheet program. See Mesh Morpher/Optimizer File Formats (p. 2867) for details about the format of the text file.

The control points will be displayed in green in the graphics window, and will turn red when selected in the **Control Points** selection list. When a control point is selected, you also have the ability to **Delete** it, or enter new coordinates and **Modify** it.

When adding control points, remember that it may be helpful to locate them not only in areas that you want to deform, but also on features in the deformation region that you want to preserve (as you can make nodes stationary by applying a parameter value of zero). When you are done adding control points, click **OK** to close the **Define Control Points** dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 35.23: Displaying the Control Points for an Unstructured Distribution



A new default name will be automatically entered in the text-entry box at the top of the **Name** group box, in preparation for any deformation regions you want to create in the future.

i. Create any additional deformation regions as necessary by repeating steps 5.a.–h.

Important

Overlapping deformation regions are not supported.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

j. If you need to modify a deformation region at any point, select it in the **Name** selection list, revise the appropriate settings, and click the **Modify** button. The updated control points and bounding box will be displayed in the graphics window.

> **Important**
>
> When modifying an existing deformation region, be sure to click **Modify** rather than **Create**. If you click **Create**, you will not modify the deformation region, but will instead create a new one with the modified settings.

k. If you need to delete a deformation region at any point, select it in the **Name** selection list and click the **Delete** button.

6. You have the option of defining constraints on the boundary zones, in order to limit the freedom of particular zones that fall within the deformation region(s) during the morphing of the mesh. The following options are available:

- unconstrained

  This option specifies that the boundary zone is completely free to be deformed according to the assigned parameters. By default, all wall zones are unconstrained.

- fixed

  This option specifies that the boundary zone is fixed and will not be deformed. None of the zones are fixed by default.

> **Important**
>
> If you specify one or more fixed boundary zones in a deformation region, you must ensure that there is at least one unconstrained boundary zone in the region as well.

- passive

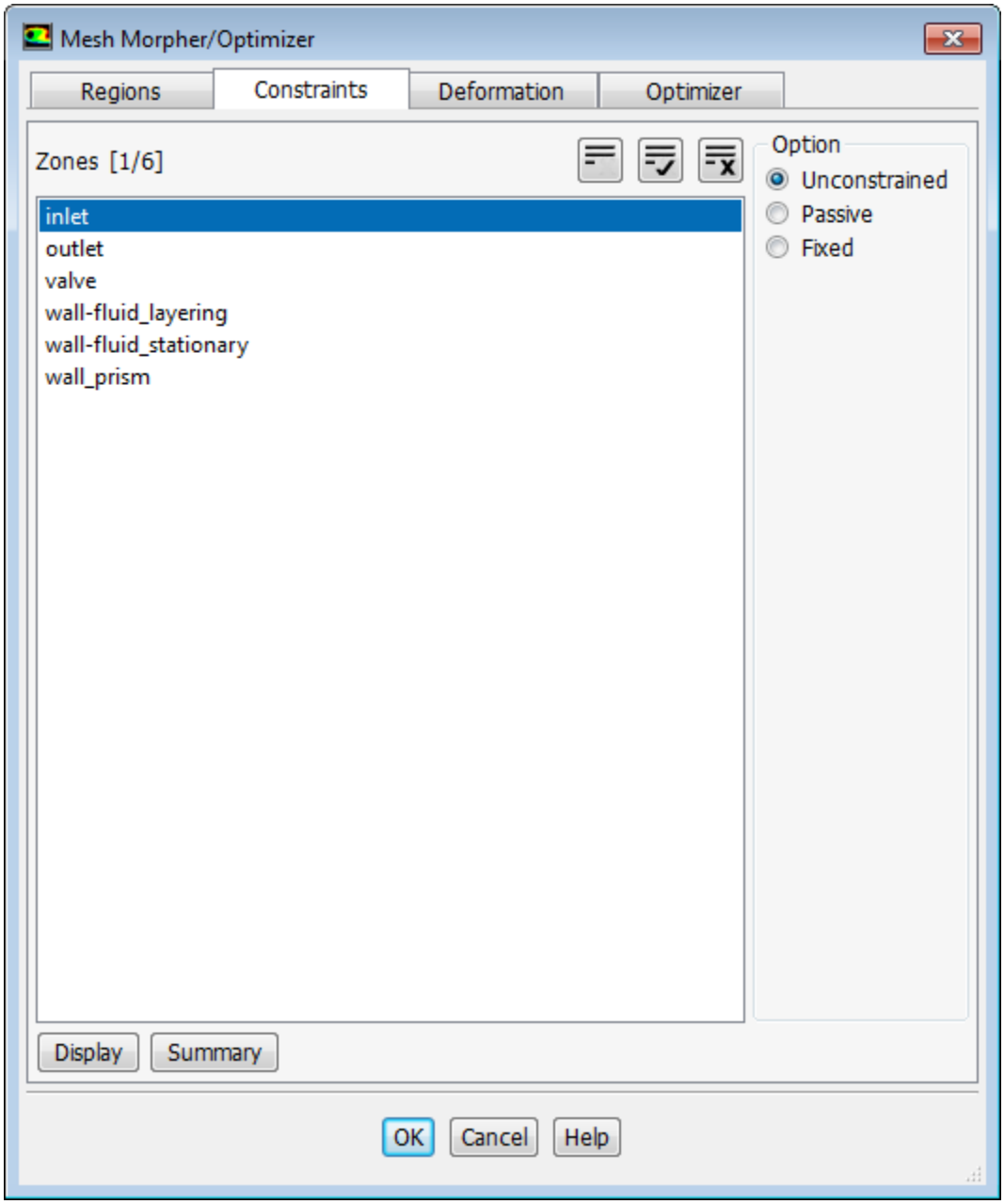
  This option specifies that the nodes of the boundary zone are partially constrained to varying degrees, based on their proximity to adjacent boundary zones that are fixed. The nodes in a passive boundary zone behave in a similar manner to the interior mesh nodes, in order to ensure that there is a smooth transition between fixed and unconstrained boundary zones. By default, all boundary zones that are not walls (for example, inlets, outlets, symmetry, and periodic boundaries) are passive.

To define the constraints on the boundary zones, perform the following steps.

a. Click the **Constraints** tab (Figure 35.24: The Constraints Tab of the Mesh Morpher/Optimizer Dialog Box (p. 2136)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 35.24: The Constraints Tab of the Mesh Morpher/Optimizer Dialog Box



b. To revise the default constraints on the boundary zones, select the zone name(s) from the **Zones** selection list in the **Constraints** group box, and then select either **Unconstrained**, **Passive**, or **Fixed** from the **Option** list.

c. Click **Display** if you want view the boundary zones selected from the **Zones** selection list, in order to verify the zones for which you are defining constraints.

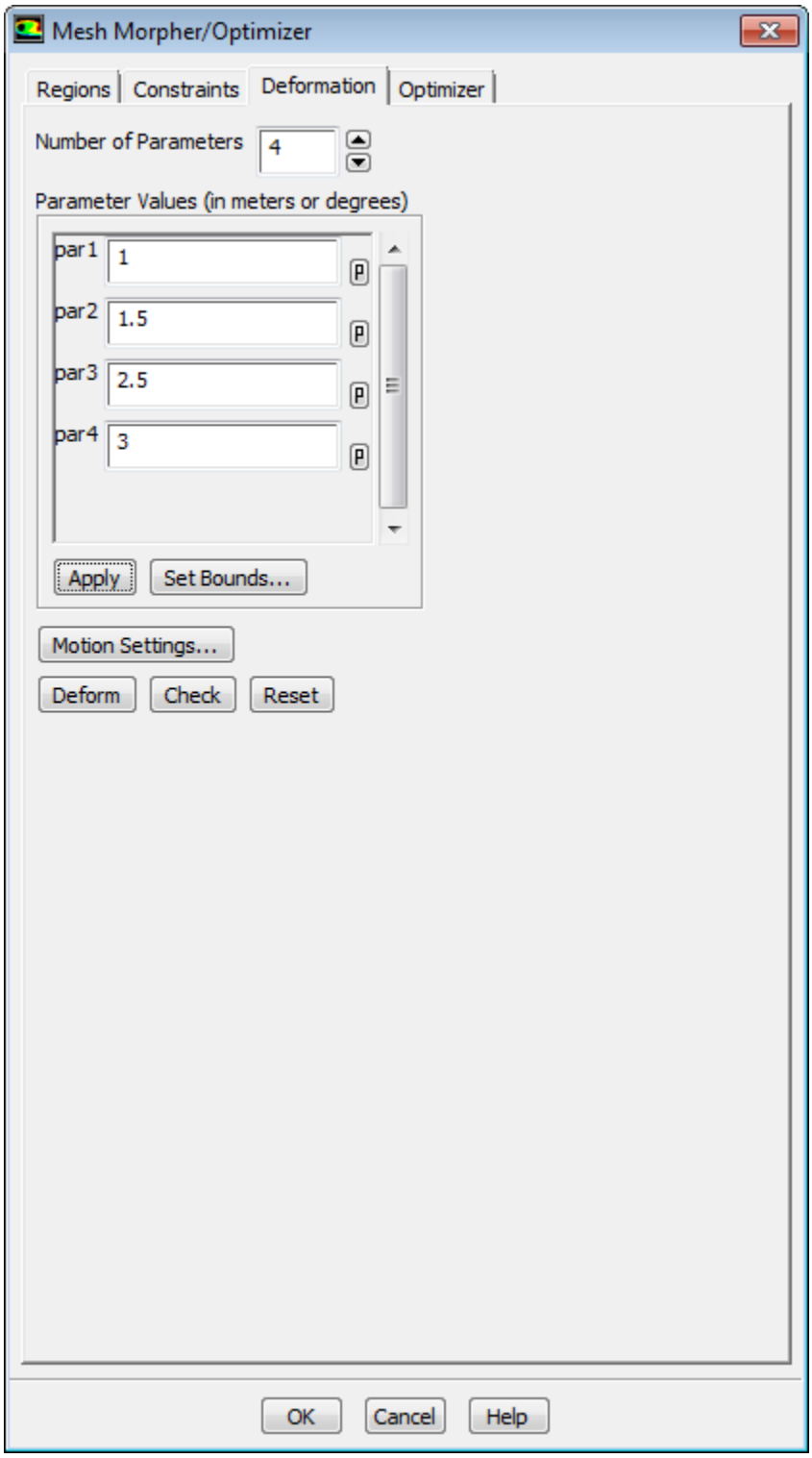
d. Click **Summary** if you would like to print a list in the console that summarizes the constraint definitions for all of the boundary zones.

7. Begin the deformation definition by defining the parameters.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Click the **Deformation** tab (Figure 35.25: The Deformation Tab of the Mesh Morpher/Optimizer Dialog Box (p. 2137)).

Figure 35.25: The Deformation Tab of the Mesh Morpher/Optimizer Dialog Box



b. Enter the **Number of Parameters** that will be used to define the deformation. This number may be as low as the maximum number of parameters you will define on a single control point, or as high as the total number of parameters you will set on all of the control points combined.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. If you want to manually specify the deformation or use Design Exploration in ANSYS Workbench to explore multiple deformation scenarios (rather than using the built-in optimizers), define the parameter fields in the **Parameter Values** group box. Note that these fields are only available when **none** or **workbench** is selected from the **Optimizer** drop-down list in the **Optimizer** tab. The value of the parameter defines a magnitude that will then be used along with other direction settings to define motions that produce an overall displacement for a given control point. The value will specify a length in meters for translations and radial motions, and will specify an angle in degrees for rotations; the units here will be used regardless of what units you are using in the case.

   You have the following options:

   - If you plan to manually specify the deformation, enter a numeric value for each field in the **Parameter Values** group box. Note that having a parameter with a value of zero allows you to make a control point stationary.

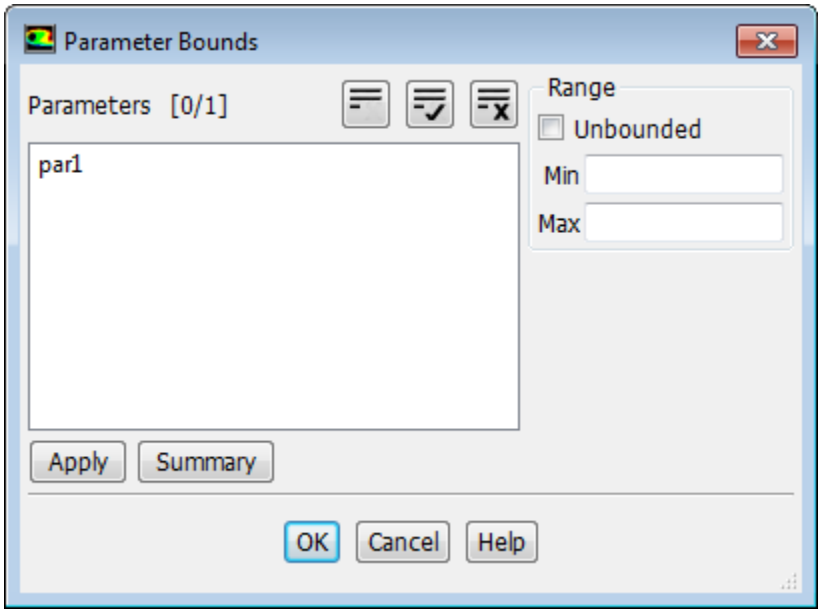
   - If you plan to use Design Exploration in ANSYS Workbench, click the P icon next to each parameter field and use the **Select Input Parameter** dialog box that opens to create or assign an input parameter with an initial value; see Select Input Parameter Dialog Box (p. 2384) for details.

   When you have finished defining the parameter fields, click the **Apply** button in the **Parameter Values** group box to save the definitions.

d. You have the option of limiting how much each parameter is allowed to deform by defining strict minimum and maximum values. This can be useful when you need to keep the parameters within a certain tolerance of the initial design, or when you know in advance that certain designs are impractical. While this option is typically used with the built-in optimizers, it is available for manual deformation, so that you can visualize the effects of the bounds and set them interactively. Note that this option is not available when **workbench** is selected from the **Optimizer** drop-down list in the **Optimizer** tab.

   Click the **Set Bounds...** button in the **Parameter Values** group box to open the **Parameter Bounds** dialog box (Figure 35.26: The Parameter Bounds Dialog Box (p. 2138)).

Figure 35.26: The Parameter Bounds Dialog Box



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

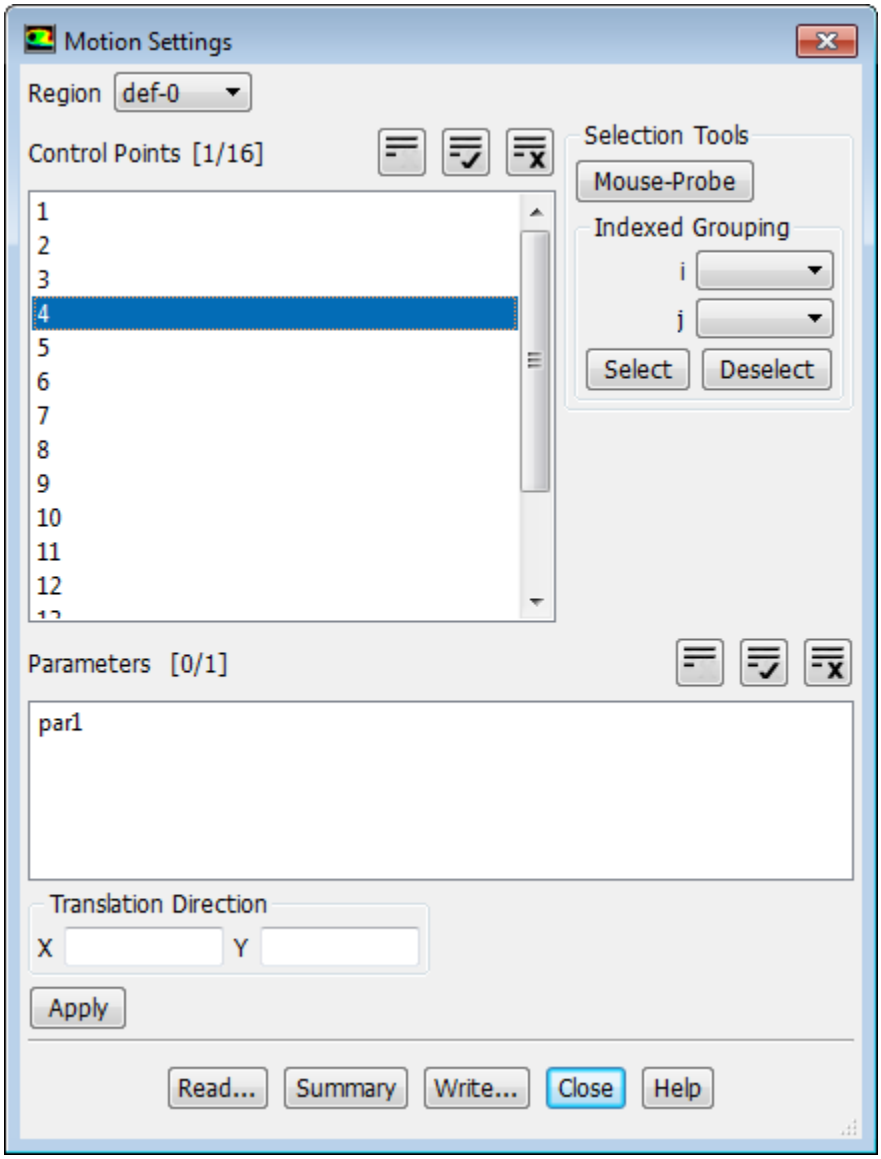
Select the parameters you want to limit from the **Parameters** selection list, disable the **Unbounded** option in the **Range** group box, and then enter appropriate values for **Min** and **Max**.

At any point you can click the **Apply** button to save your bounds, and you can click the **Summary** button to display a summary of the saved bounds in the console. When all of the parameter bounds are defined to your satisfaction, click the **OK** button to close the **Parameter Bounds** dialog box.

8. If you selected **Regular** for the **Control Point Distribution** in the **Regions** tab, perform the following steps. You will define motion settings for the control points by selecting parameters (which you defined previously) and specifying the translation direction, thus completing the deformation definition.

   a. Click the **Motion Settings...** button in the **Deformation** tab to open the **Motion Settings** dialog box (Figure 35.27: The Motion Settings Dialog Box for a Regular Distribution (p. 2139)).

Figure 35.27: The Motion Settings Dialog Box for a Regular Distribution



b. (optional) If you have previously generated an ASCII text file that contains motion settings, you can read it by clicking **Read...** at the bottom of the **Motion Settings** dialog box and using the dialog box that opens; otherwise, you should proceed to steps 8.c.–8.j. and define the settings using the other controls in the **Motion Settings** dialog box.

Using a text file can be helpful when you have a large number of motion settings. It is not necessary to create the text file from scratch, as you can create a simplified text file with sample values using the **Write...** button (as described in step 8.k.), and then add the bulk of the motion settings using a spreadsheet program. See Mesh Morpher/Optimizer File Formats (p. 2867) for details about the format of the text file.

Important

Immediately after you read a text file, the **Motion Settings** dialog box will not display the applied settings. To begin viewing the applied settings, make a selection from the **Region**, **Control Points**, and **Parameters** lists, and the associated **Translation Direction** will be displayed.

You can then edit the motion settings as necessary (in a manner similar to steps 8.c.–8.g.) and/or proceed to step 8.m.

c. Make a selection from the **Region** drop-down list in the **Motion Settings** dialog box.

d. Specify the control points in the current **Region** to which you want to assign the same motion settings (that is, the same combination of parameters and direction). You can specify these control points using one or more of the following methods:

- Select the ID numbers from the **Control Points** selection list. The selected control points will be highlighted in the graphics window.
- Click the **Mouse-Probe** button in the **Selection Tools** group box and then click the control points in the graphics window with the mouse-probe button (which is the right mouse button, by default). To deselect, select a highlighted control point with the mouse-probe button. As you select the control points, they will also be selected in the **Control Points** selection list.
- Use the **Indexed Grouping** group box to select the control points based on their assigned indices. Each control point has **i**, **j**, and (for 3D cases) **k** index numbers, each of which denote its place in the sequence (starting at the origin of the region) of control points along the direction-1, direction-2, and direction-3 vectors, respectively (as defined in the **Regions** tab of the **Mesh Morpher/Optimizer** dialog box). You can make selections from the index drop-down lists, and then click the **Select** button. In a similar way you can deselect control points using the **Deselect** button. As you select the control points, they will also be selected in the **Control Points** selection list and highlighted in the graphics window.

  For example, consider the case of a 3D region: if you select **2** from the **k** drop-down list, the control points in the second plane perpendicular to the direction-3 vector (that is, the direction-1–direction-2 plane) will be selected. If you select **1** from both the **i** and **j** drop-down lists, the line of control points at the intersection of the first planes perpendicular to the direction-1 and direction-2 vectors will be selected. Making selections from all three drop-down lists will select a single control point.

e. Make selections from the **Parameters** selection list, in order to specify those you want to assign (along with a translation direction) to the selected **Control Points**. Note that the values associated with these

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

parameters (as displayed in the **Parameter Values** group box in the **Deformation** tab of the **Mesh Morpher/Optimizer** dialog box) are irrelevant if you plan to use the built-in optimizers.

f. Define the **Translation Direction** that will be associated with the selected **Parameters** by entering values for the **X**, **Y**, and (for 3D cases) **Z** directions. Note that you do not need to define a unit vector.

   If you use a built-in optimizer, these values will provide the direction of the displacement of the control point and the optimizer will determine the overall magnitude of displacement. Alternatively, if you specify the deformation manually or using Design Exploration, the values you enter in the **Translation Direction** group box will be multiplied with the values of the **Parameters** to define the displacement applied to the control points.

g. Click the **Apply** button to save the motion settings.

h. If you want to apply another set of motion settings to the selected **Control Points**, repeat steps 8.e.–8.g. for each additional combination of parameters and direction.

i. Repeat steps 8.d.–8.h. for each additional set of control points in the current **Region** to which you want to assign motion settings.

j. Repeat steps 8.c.–8.i. for each additional region in which you want to apply deformation parameters to control points.

k. If you want to save all of the settings you specified in the **Motion Settings** dialog box as an ASCII text file, click **Write...** and specify a name in the dialog box that opens. You can edit the motion settings in this text file using a spreadsheet program, and then read it in this or a separate case file, as described in step 8.b.

l. You can display a summary of the motion settings in the console by clicking the **Summary** button.

m. Click **OK** to close the **Motion Settings** dialog box.

9. If you selected **Unstructured** for the **Control Point Distribution** in the **Regions** tab, perform the following steps. You will define one or more motions—which consist of a parameter (defined previously), direction settings, and the affected control points—and thus complete the deformation definition.

Important

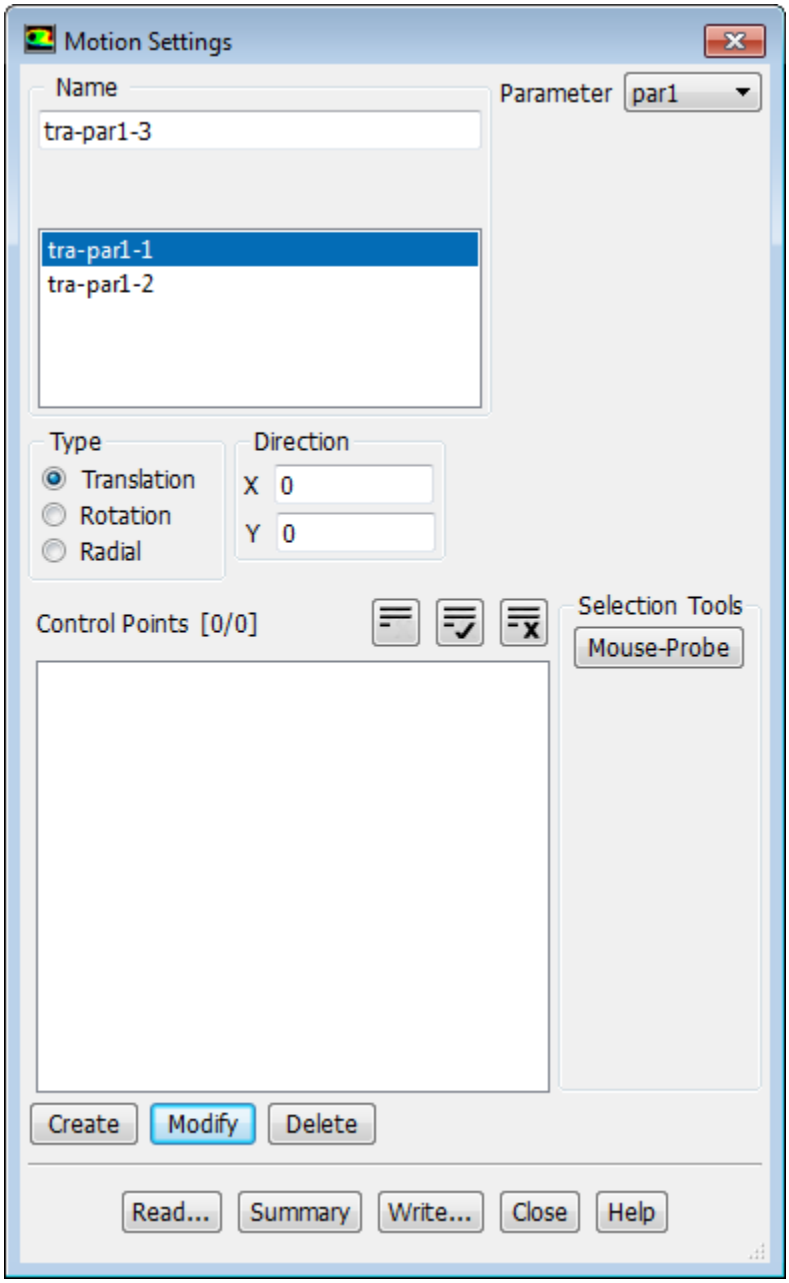
When a single control point is associated with multiple motions, note that the resultant displacement for each individual motion will be calculated from the original position, and then the displacements will be summed. This means that the order in which the motions are created will have no effect on the final location of the control point.

---

   a. Click the **Motion Settings...** button in the **Deformation** tab to open the **Motion Settings** dialog box (Figure 35.28: The Motion Settings Dialog Box for an Unstructured Distribution (p. 2142)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 35.28: The Motion Settings Dialog Box for an Unstructured Distribution



b. (optional) If you have previously generated an ASCII text file that contains motion definitions, you can read it by clicking **Read...** at the bottom of the **Motion Settings** dialog box and using the dialog box that opens; otherwise, you should proceed to steps 9.c.–9.k. and define the settings using the other controls in the **Motion Settings** dialog box.

   Using a text file can be helpful when you have a large number of motions. It is not necessary to create the text file from scratch, as you can create a simplified text file with sample values using the **Write...** button (as described in step 9.l.), and then add the bulk of the motion definitions using a spreadsheet program. See Mesh Morpher/Optimizer File Formats (p. 2867) for details about the format of the text file.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can then either add additional motions (in a manner similar to steps 9c.–9.h.) or modify / delete an existing motion (as described in steps 9.i. and 9.j.). When the settings are finalized, you can proceed to step 9.k.

c. To begin defining a new motion, make a selection from the **Parameter** drop-down list.

d. Make a selection from the **Type** list, to indicate the type of motion you would like to define. You can specify that a group of control points undergoes a **Translation**, a **Rotation** about a point (for 2D) or an axis (for 3D), or a **Radial** motion about a point (for 2D) or an axis (for 3D).

e. Define the motion by entering **X**, **Y**, and (for 3D cases) **Z** values for the following: the **Direction** used for translations; or the **Axis Origin** and (for 3D cases) **Axis Direction** used for rotations and radial motions. For **Direction** or **Axis Direction**, note that you do not need to define unit vectors.

   If you use a built-in optimizer, the values you enter will define the direction of the displacement of the control point and the optimizer will determine the overall magnitude of displacement. Alternatively, if you specify the deformation manually or using Design Exploration, the magnitude will be the value of the **Parameter**; translation is a special case, in which the values you enter in the **Direction** group box are multiplied with the value of the **Parameter** to define the displacement applied to the control points.

f. Select the control points to which you want to assign the selected motion. You can either select them in the **Control Points** list, or click the **Mouse-Probe** button and click the control points in the graphics window using the mouse-probe button (which is the right mouse button, by default).

g. A default name will be generated in the top field of the **Name** group box, based on the **Type** and **Parameter** of the motion. You can edit this text to make it more descriptive, as necessary.

h. Click **Create** to create the motion. The name will be added to the selection list in the **Name** group box.

i. Repeat steps 9.c.–9.g. for each additional motion that you would like to create.

j. If you need to modify a motion at any point, select it in the lower selection list in the **Name** group box, revise the appropriate settings, and click the **Modify** button.

Important

When modifying an existing motion, be sure to click **Modify** rather than **Create**. If you click **Create**, you will not modify the motion, but will instead create a new one with the modified settings.

---

k. If you need to delete a deformation region at any point, select it in the lower selection list in the **Name** group box and click the **Delete** button.

l. You can display a summary of the motion definitions in the console by clicking the **Summary** button.

m. If you want to save all of the settings you specified in the **Motion Settings** dialog box as an ASCII text file, click **Write...** and specify a name in the dialog box that opens. You can edit the motion settings in this text file using a spreadsheet program, and then read it in this or a separate case file, as described in step 9.b.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10. If you want to manually specify the deformation (that is, **none** is selected from the **Optimizer** drop-down list in the **Optimizer** tab of the **Mesh Morpher/Optimizer** dialog box), perform the following steps:

    a. Click the **Deform** button to apply the current settings (in the **Parameter Values** group box and **Motion Settings** dialog box) and to display the deformed mesh in the graphics window. The **Deform** button allows you to manually specify the deformation.

    b. You can click the **Check** button to print out a mesh check report in the console for the currently displayed mesh. The mesh check report is the same as that produced by the **Check** button in the **General** task page, as described in Mesh Check Report (p. 193).

    Important

    If you save parameter settings but do not click the **Deform** button, the mesh check report will not account for the mesh that is produced from those settings.

    c. Click the **Reset** button if you want to revert to the original mesh without the deformations that result from the **Deform** button.

11. If you plan to use the built-in optimizers, you should decide whether you want to disable the general mesh check that is performed by default immediately after the mesh is deformed in every design stage. The mesh check that is performed is the same as that initiated by the **Check** button in the **General** task page (see Checking the Mesh (p. 192) for details). If any errors are discovered, the mesh is rejected and the next design stage is attempted.

    Disabling the general mesh check allows you to repair the mesh, so that an accurate solution can be calculated for it. Note that you can set up the mesh repair by entering the appropriate text commands (as described in Repairing Meshes (p. 194)) in the **Initial Commands** text-entry box in the **Optimizer** tab.

    You can disable the general mesh check by using the following text command:

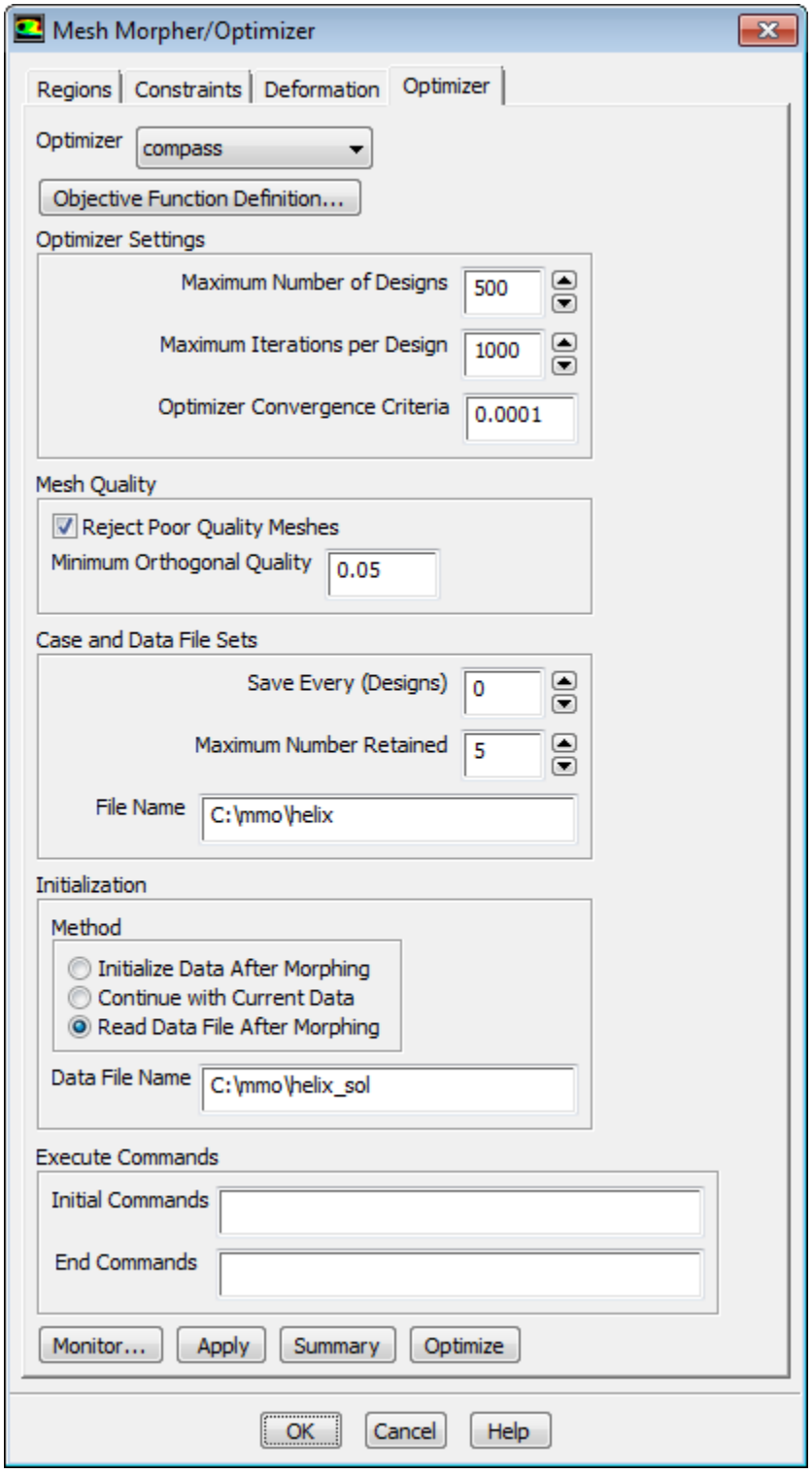
    `define → mesh-morpher-optimizer → optimizer-parameters → disable-mesh-check`

12. If you want to use the built-in optimizers, define the optimizer settings and initiate the optimization process.

    a. Click the **Optimizer** tab (Figure 35.29: The Optimizer Tab of the Mesh Morpher/Optimizer Dialog Box (p. 2145)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 35.29: The Optimizer Tab of the Mesh Morpher/Optimizer Dialog Box

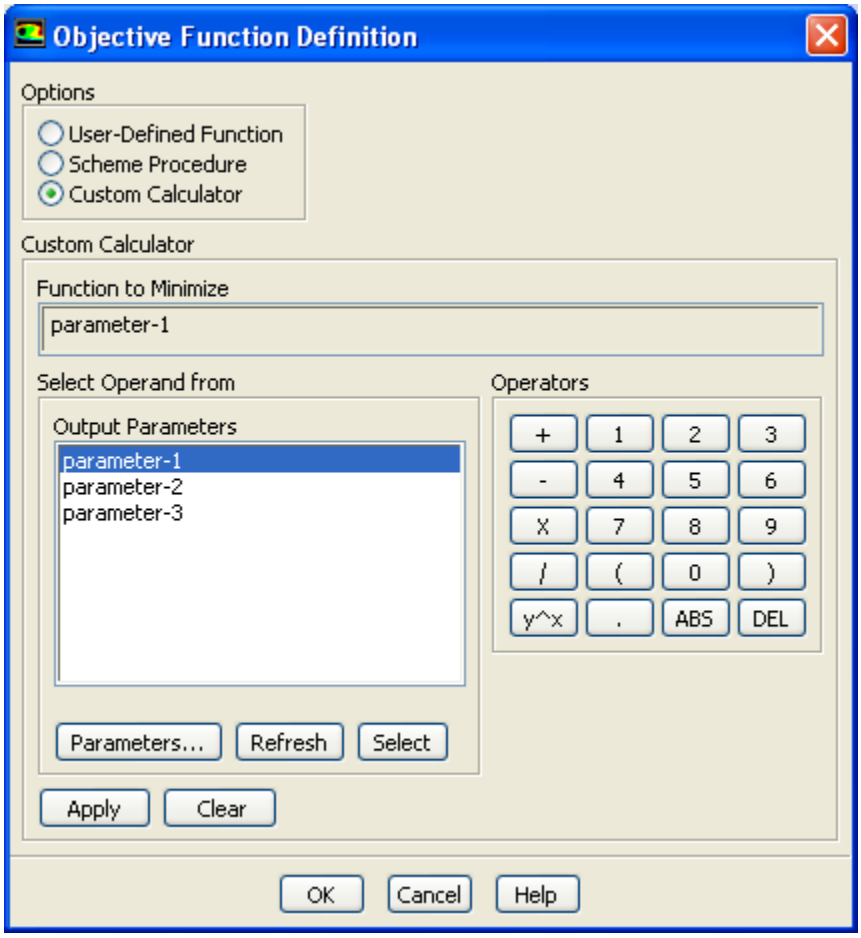


b. Select an optimizer from the **Optimizer** drop-down list. The available optimizers are not based on gradients, and include the following: **compass**, **newuoa**, **powell**, **rosenbrock**, **simplex**, and **torczon**. For more information about how these optimizers function, see Optimizers (p. 2123).

c. Click the **Objective Function Definition...** button to open the **Objective Function Definition** dialog box (Figure 35.30: The Objective Function Definition Dialog Box (p. 2146)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 35.30: The Objective Function Definition Dialog Box



i. Make a selection in the **Options** group box to specify whether the objective function that will be minimized during the optimization process is a **User-Defined Function**, a **Scheme Procedure**, or a customized function of output parameters as defined by the **Custom Calculator**. Your selection should correspond with the actions you took in step 2.

ii. If you selected **Custom Calculator** in the previous step, define the objective function via the GUI controls in the **Custom Calculator** group box. As you click the buttons in this group box, text and symbols will appear in the **Function to Minimize** text box. You *cannot* edit the contents of this box directly; if you want to delete part or all of the function, use the **DEL** or **Clear** button, respectively.

You can include output parameters in the definition of the function by making a selection from the **Output Parameters** list and clicking **Select**. If you want to add a new output parameter to the selection list or check the definition of an existing one, you can click the **Parameters...** button and use the **Parameters** dialog box (as described in Creating Output Parameters (p. 1909)); be sure to click the **Refresh** button when you return to the **Objective Function Definition** dialog box, so that the **Output Parameters** list is updated. You can also use **Operators** to manipulate the output parameters in the function.

You can click the **Apply** button at any point to save your function. When it is fully defined, click the **OK** button to save your settings and close the **Objective Function Definition** dialog box.

d. Define the settings in the **Optimizer Settings** group box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

i. Enter a value for **Maximum Number of Designs**, to specify the maximum number of design stages the optimizer will undergo to reach the specified objective function. Note that this number is not the number displayed in the console during the optimization process. Each optimizer undergoes a certain number of design modifications (called "runs") before it converges to a design for a particular design stage. The Run number displayed in the console refers to the actual design modifications (including the inner loop) irrespective of the optimizer, whereas the **Maximum Number of Designs** value is the maximum number of converged designs provided by the optimizer.

ii. Enter the **Maximum Iterations per Design**, to specify the maximum number of iterations ANSYS Fluent performs for each design change.

iii. Enter a number for **Optimizer Convergence Criteria**, to specify the convergence criteria for the optimizer.

iv. If you selected the **newuoa** optimizer, enter a number for the **Initial Parameter Variation** to specify how much the parameters will be allowed to vary during the initial calculations. This value is not intended to define a strict minimum / maximum for the parameters (as is defined by the bounds specified using the **Parameter Bounds** dialog box), but should only be large enough to allow the optimizer to capture the minima. A better estimation (that is, a smaller value) allows the optimizer to reach convergence faster.

e. By default, the orthogonal quality (as defined in Mesh Quality (p. 131)) of the initial and every subsequent mesh is computed, before the related solution calculation is initiated. If the orthogonal quality for any cell is less than a specified value, the mesh is rejected and Fluent proceeds to the next design stage (therefore helping to ensure that your solution accuracy is not compromised by poor quality meshes). These actions are governed by the settings in the **Mesh Quality** group box.

If you do not want the meshes rejected based on their orthogonal quality, you can disable the **Reject Poor Quality Meshes** option; otherwise, specify the minimum orthogonal quality that must be maintained by every cell by entering a value from `0`–`1` (where `0` represents the worst quality) for **Minimum Orthogonal Quality**. When determining an appropriate minimum value, you can check the orthogonal quality of the initial mesh (by using the **Report Quality** button in the **General** task page) and then select a value that is sufficiently lower, so as not to reject a majority of the meshes generated during the optimization process.

f. You have the option of saving intermediate case and data files during the optimization run, so that you can restart an interrupted solution in the same or a different Fluent session without increasing the overall number of design iterations needed to reach convergence. To enable the saving of such intermediate files, define the settings in the **Case and Data File Set** group box.

i. Specify the frequency (in number of design iterations) that you want the intermediate case and data files saved by entering a value for **Save Every**. The default value is **0**, which specifies that no intermediate files will be saved. Note that if you selected the **newuoa** optimizer, the frequency at which files are saved may not exactly correspond to the number you enter, but should be reasonably close.

ii. Specify the maximum number of intermediate files sets you want to retain by entering a value for **Maximum Number Retained**. After the maximum limit of file sets has been saved, ANSYS Fluent begins overwriting the earliest existing intermediate file set. Lower values should be used if you have limited disk space or you are concerned about saving unnecessary files.

iii. Specify a root name for the intermediate files in the **File Name** text box. When the files are saved, the design iteration will be appended to this root name to indicate the point at which it was saved during the optimization run. An extension will also be automatically added to the root name (`.cas`

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

or `.dat`). You can include a folder path in the filename if you want the files saved outside of the working folder.

g. Specify how the solution variables should be treated after the mesh is deformed by making a selection from the **Method** list in the **Initialization** group box:

- Select **Initialize Data After Morphing** to specify that the solution variables should be initialized to the values specified in the **Solution Initialization** task page after deformation.
- Select **Continue with Current Data** to specify that the solution variables remain the values obtained in the previous design stage. This selection will reduce the number of iterations needed to reach convergence compared to the **Initialize Data After Morphing** selection. Note that if the solver diverges during an intermediate design stage, the solution variables will be initialized and the solution will be attempted again.
- Select **Read Data File After Morphing** to specify that the solution variables are set to the values obtained from the data file specified in the **Data File Name** text-entry box. This selection will reduce the number of iterations needed to reach convergence compared to **Initialize Data After Morphing** selection or (in cases where the solution diverges for an intermediate design stage) the **Continue with Current Data** selection. Note that this selection is not available when **newuoa** is selected from the **Optimizer** drop-down list.

h. If you did not select the **newuoa** optimizer, you have the option of specifying commands that will be executed during the optimization runs of the mesh morpher/optimizer, via the text-entry boxes in the **Execute Commands** group box. A command can be a text command or the name of a command macro you have defined (or will define), as described in Defining Macros (p. 1640). You can also enter a series of text commands and/or macros, as long as they are separated by a semi-colon (`;`).

During optimization runs, deformation occurs as part of every design iteration. You decide the specific point during the design iteration that the command is executed:

- If you want a command to be executed after the design has been modified, but before ANSYS Fluent has started to run the calculation for that design stage, enter the command in the **Initial Commands** text-entry box. There are no restrictions on the commands that can be entered: you can enter commands that read saved data, perform FMG initialization, execute an entirely independent on-demand UDF, or even call a Scheme routine.
- If you want a command to be executed after the solution has run and converged for a design stage, enter the command in the **End Commands** text-entry box. There are no restrictions on the commands that can be entered. Examples are commands for postprocessing solution variables, monitoring contours and vectors of different variables, taking snapshots of each design change, and so on, at every design stage.

Important

If the command to be executed involves saving a file, see Saving Files During the Calculation (p. 1642) for important general information.

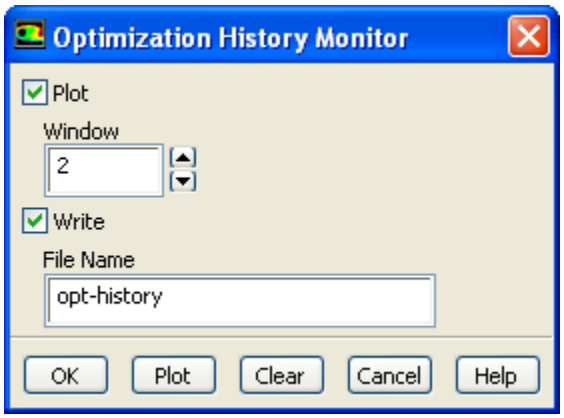
As noted in Automatic Numbering of Files (p. 43), the special character, `%i`, can be used to create unique filenames by including the iteration number. However, by default, the solution will be initialized after each mesh deformation and the iteration count restarted. Therefore, the resulting filename cannot be used to identify which design stage produced it (that is, the first design stage may con-

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

verge at 50 iterations, and then the second design stage may converge at 43 iterations). The time that the file was created is a better way to identify where in the series of design stages the file was generated. In addition, some files may be overwritten if two design stages converge in the same number of iterations. Alternatively, if you have selected **Continue with Current Data** under **Initialization** when setting up the optimizer, then the iteration count will not reset and no duplicate filenames will occur.

i. If you did not select the **newuoa** optimizer, you can plot and/or record the optimization history (that is, how the value of the objective function varies with each design stage). Click the **Monitor...** button to open the **Optimization History Monitor** dialog box (Figure 35.31: The Optimization History Monitor Dialog Box (p. 2149)) and perform the steps that follow.

Figure 35.31: The Optimization History Monitor Dialog Box



i. Enable the **Plot** option if you want to display a plot of the optimization history in the graphics window indicated in the **Window** number-entry box. Note that if you want to observe the mesh deformations during the optimization process, you must make sure that **Window** is not set to the ID of the graphics window that is displaying the mesh.

ii. Enable the **Write** option if you want to save the optimization history data in a file, and specify the **File Name**.

Note that you can display a plot of the optimization history data generated during the last calculation, even if the **Plot** or the **Write** options were not enabled during the calculation, as long as you have not discarded the data using the **Clear** button. Simply click the **Plot** button and the plot will be displayed in the active graphics window.

j. Click the **Apply** button to save your optimizer settings.

k. Click the **Summary** button to display a summary of the mesh morpher/optimizer settings in the console.

l. Click the **Optimize** button to initiate the optimization process. Information about each run (that is, each design modification) that is generated as part of the production of a design stage will be displayed in the console, and the deformed mesh for each run will be updated in the graphics window.

If you set up the saving of intermediate case and data files in step 12.f and your solution is interrupted during the optimization run, you can restart the calculation using the latest intermediate files. Note that you should not revise the deformation regions or constraint settings for the restarted mesh.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13. If you want to use Design Exploration in ANSYS Workbench to explore multiple deformation scenarios, define the optimizer settings.

    a. Click the **Optimizer** tab (Figure 35.29: The Optimizer Tab of the Mesh Morpher/Optimizer Dialog Box (p. 2145)).

    b. Select **workbench** from the **Optimizer** drop-down list. Note that **workbench** is only available if you have launched your ANSYS Fluent session from ANSYS Workbench.

    c. If necessary, revise the default settings in the **Mesh Quality** group box, as described in step 12.e. Note that the design point associated with a poor quality mesh will not be updated in Workbench.

14. Click **OK** to close the **Mesh Morpher/Optimization** dialog box.

15. If you want to use Design Exploration in ANSYS Workbench to explore multiple deformation scenarios, save your case file and close your ANSYS Fluent session. You can then proceed to ANSYS Workbench and define the design points that will provide the multiple values for the input parameters you created / assigned in step 7.c. For more information about using design points to explore ANSYS Fluent results, see Working With Input and Output Parameters in Workbench in the ANSYS Fluent in Workbench User's Guide.

16. If you manually deformed the mesh or used the built-in optimizers, run your case file with the deformed mesh.

17. If you want to rerun a case file in which the mesh has been deformed by the **Optimize** button in the **Optimizer** tab, you should not revise the deformation regions or constraint settings for the deformed mesh.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 36: Customizing Fluent

36.1. Customizing Fluent with ANSYS ACT

ANSYS ACT functionality is available for stand-alone Fluent as well as Fluent in Workbench. There are three ways that you can enable ACT functionality:

- Enable the **Load ACT** option in the Fluent Launcher.
- Select **ACT** from the **Arrange the workspace** ( ) drop-down.
- Enter the text command: `file/load-act-tool`.

The **ACT Start Page** *will open as a new component in the Fluent layout*.

36.1.1. ACT Start Page: Accessing ACT Functionality

The **ACT Start Page** provides an access point for ACT functionality. From this page, you can access tools that can be used in the development and execution of extensions.

For additional information, see Using the ACT Start Page in the *ANSYS ACT Developer's Guide*.

36.1.2. Extension Manager: Working with Extensions

Click the **Extension Manager** icon on the **ACT Start Page**.

The **Extension Manager** tool shows all of the extensions that are currently installed (scripted extensions are listed only if you have an ACT license). Extensions with green backgrounds are the ones that have already been loaded. The following general functionality is available:

- Click a "+" icon to install a new binary extension.
- Click the gear icon to add a directory that ACT will search for extensions. (This is how you will install scripted extensions since the install extension capability applies only to binary extensions.)

> **Note**
>
> The directory containing the IronPython script must have the same name as the `.xml` extension file.

- Click the **ANSYS App Store** button to access extensions available for download.
- Use the search field to find a specific extension.

You can also right-click an extension to open a context menu showing the available options for that extension, including:

- Loading/unloading extensions
- Specifying whether an extension is loaded by default
- Uninstalling binary extensions
- Building scripted extensions

For more detailed information, see Extension Manager Accessed via the ACT Start Page in the *ANSYS ACT Developer's Guide*.

36.1.3. Wizards Page: Working with Wizards

ACT wizards enable you to use ACT's process-compression and automation capabilities in Fluent, regardless of whether Fluent is opened in Workbench or as a stand-alone application. Once launched, a wizard provides guidance within the application workflow, walking users step-by-step through a fluid dynamics simulation.

An ACT wizard is encapsulated in an extension, placed within a `<wizard>` block in the extension XML definition file; one or more wizards can be defined within the same extension. In order to use a wizard, you must install and load the extension via the Extension Manager, as described in Extension Manager: Working with Extensions (p. 2151).

To access the **Wizards** page, click the corresponding icon on the **ACT Start Page**. The **Wizards** tool shows all of wizards currently installed for Fluent. You can use the **Search** field to search for a wizard by the name, description, or author of the extension.

To execute a wizard, click it. Alternatively, you can select **Execute Wizard** from the wizard's context menu. The wizard interface will open, showing the first step of the wizard.

For more information on wizards in general, see Using Wizards in the *ANSYS ACT Developer's Guide*.

36.1.3.1. Fluent Wizard Example: Simple Analysis

This example uses a Fluent wizard called **SimpleAnalysis**. It is defined in the extension **FluentDemo** (XML Extension Definition File (p. 2152)). You can find this extension in the **ACT Templates** section of the ANSYS Customer Portal.

This example imports a mesh, runs a simple analysis, generates results, and displays the results in a customized panel in the UI. At minimum, an extension consists of an XML Extension Definition File (p. 2152) and an IronPython Script (p. 2155).

Note

This example requires you to run Fluent in 3D (selected from the Fluent Launcher). It also requires you to download the mesh file `final.msh` from the ANSYS App Store. When you click **ACT Wizard Templates** under **ACT Templates**, you will be prompted to download the folder.

36.1.3.1.1. XML Extension Definition File

This section contains the `FluentDemo.xml` extension file, as well as explanations of each of the required fields.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Extension Definition

The `name` and `version` attributes define the extension name and version. The `guid` specifies a unique identifier for the extension.

Script Reference

The `<script>` block specifies the IronPython script referenced by the extension. This wizard references the script `main.py` (IronPython Script (p. 2155)).

Interface Definition

The `<interface>` block defines details of the interface.

- The `context` attribute is set to `Fluent` because the extension is executed in the Fluent application.
- The `<images>` block refers the extension **images** folder; the image contained in this folder is displayed as the icon for the wizard.

User Interface Definition

The `<uidefinition>` block allows you to customize the ACT user interface, by creating specialized views and layouts.

- The `<layout>` block allows you to define a custom layout. In this example, only a single layout is defined.
  - The `name` attribute is set to `ResultsView`, specifying the name of the layout being defined (a customized view for the display of Fluent results).
  - The following child `<component>` blocks define the details and display properties of each UI component included in the custom layout: `Title`, `Properties`, `Report`, `Chart`, and `Submit`.

Wizard Definition

The wizard and its steps are defined in the `<wizard>` block.

- The required `name` and `version` attributes define the wizard name and version.
- The `context` attribute is set to `Fluent` because the wizard will be executed in the Fluent application.
- The `icon` attribute references the `wizard_icon.png` image in the **images** folder.

Step Definitions

The `<step>` tab is used to define the steps of the wizard. There are two steps:

`Analysis` and `Results`.

For each step:

- The `name`, `version`, and `caption` attributes define the step name, version, and display text for the step.
- The `<callbacks>` block defines the callbacks to the functions defined in the `main.py` Python script.
  - For the Analysis step:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ The <onrefresh> callback executes the ImportMesh function.

→ The <onupdate> callback executes the CreateAnalysis function.

– For the Results step, the <onrefresh> callback executes the CreateReport function.

- The <property> block for each step defines a single property and its attributes.

XML Example:

```
<extension name="FluentDemo" version="1">
 <guid>A0A3E28C-E094-4B2D-8A91-A0B0884D1AE2</guid>
 <author>ANSYS Inc.</author>
 <description>This extension demonstrates how to create a wizard in Fluent.</description>

 <script src="main.py" />

 <interface context="Fluent">
  <images>images</images>
 </interface>

  <uidefinition>

  <layout name="ResultsView">
   <component
    name="Title"
    topAttachment=""
    topOffset="10"
    leftAttachment=""
    leftOffset="10"
    rightAttachment=""
    rightOffset="10"
    bottomOffset="10"
    bottomAttachment="Properties"
    widthType="Percentage"
    width="100"
    heightType="FitToContent"
    height="200"
    componentType="startPageHeaderComponent" />
   <component
    name="Properties"
    topAttachment="Title"
    leftAttachment=""
    leftOffset="10"
    rightAttachment=""
    rightOffset="10"
    bottomAttachment="Report"
    bottomOffset="10"
    widthType="Percentage"
    width="100"
    heightType="Percentage"
    height="20"
    componentType="propertiesComponent" />
   <component
    name="Report"
    topAttachment="Properties"
    leftAttachment=""
    leftOffset="10"
    rightAttachment=""
    rightOffset="10"
    bottomAttachment="Chart"
    bottomOffset="10"
    widthType="Percentage"
    width="100"
    heightType="Percentage"
    height="40"
    componentType="helpContentComponent" />
   <component
    name="Chart"
    topAttachment="Report"
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
    leftAttachment=""
    leftOffset="10"
    rightAttachment=""
    rightOffset="10"
    bottomAttachment="Submit"
    bottomOffset="10"
    widthType="Percentage"
    width="100"
    heightType="Percentage"
    height="40"
    componentType="chartComponent" />
   <component
    name="Submit"
    topAttachment="Chart"
    leftAttachment=""
    leftOffset="10"
    rightAttachment=""
    rightOffset="10"
    bottomAttachment=""
    bottomOffset="10"
    widthType="Percentage"
    width="100"
    heightType="FitToContent"
    height="50"
    componentType="buttonsComponent" />
  </layout>

 </uidefinition>


 <wizard name="Simple Analysis" version="1" context="Fluent" icon="wizard_icon">
  <description>Generate the mesh and solve the analysis.</description>

  <!--<step name="Mesh" caption="Mesh" version="1">
   <callbacks>
    <onupdate>CreateMesh</onupdate>
   </callbacks>

   <property name="minSize" caption="Minimum element size" control="float" default="0.001" />
   <property name="maxSize" caption="Maximum element size" control="float" default="0.01" />
  </step>-->

  <step name="Analysis" caption="Analysis" version="1">
   <callbacks>
    <onrefresh>ImportMesh</onrefresh>
    <onupdate>CreateAnalysis</onupdate>
   </callbacks>

   <property name="velocity" caption="Velocity" control="float" default="3" />
  </step>

  <step name="Results" caption="Results" version="1" layout="ResultsView@FluentDemo">
   <callbacks>
    <onrefresh>CreateReport</onrefresh>
   </callbacks>

   <property name="pressure" caption="Pressure Drop" control="text" readonly="true" />
  </step>

 </wizard>

</extension>
```

36.1.3.1.2. IronPython Script

This section contains the IronPython script, `main.py`. It is located in the **FluentDemo** extension folder.

This script defines all of the functions executed by the callbacks in the XML extension file: `ImportMesh`, `CreateAnalysis`, and `CreateReport`. Each step defined in the XML file may include multiple actions.

Note

The `ImportMesh` function references the input file `final.msh`. It is located in the **Fluent-Demo** extension folder, along with the `main.py` script.

```
def CreateMesh(step):
    tui = ExtAPI.Application.ScriptByName("TUI")

    tui.SendCommand("""switch-to-meshing-mode""")
    tui.SendCommand("""/file/import/cad yes "{0}" 40 yes m""".format(r"e:\models\cyl.agdb"))
    tui.SendCommand("""/objects/separate-faces-by-angle/solid () 90""")
    tui.SendCommand("""/size-functions/set-global-controls 0.001 0.01 1.2""")
    tui.SendCommand("""/scoped-sizing/create "control-1" curvature object-faces-and-edges yes
 no "*" 0.001 0.01 1.2 18""")
    tui.SendCommand("""/scoped-sizing/create "control-2" proximity object-faces-and-edges yes
 no "*" 0.001 0.01 1.2 10 face-face no yes""")
    tui.SendCommand("""/scoped/compute""")
    tui.SendCommand("""/objects/change-object-type solid () mesh yes""")
    tui.SendCommand("""/objects/improve-object-quality solid () surface-remesh""")
    tui.SendCommand("""/boundary/manage/name solid "inlet" """)
    tui.SendCommand("""/boundary/manage/name solid:5 "outlet" """)
    tui.SendCommand("""/boundary/manage/name solid:6 "wall" """)
    tui.SendCommand("""/boundary/manage/type (*) wall""")
    tui.SendCommand("""/boundary/manage/type outlet () pressure-outlet""")
    tui.SendCommand("""/boundary/manage/type inlet () velocity-inlet""")
    tui.SendCommand("""/objects/join-intersect/compute-regions "solid" no""")
    tui.SendCommand("""/objects/join-intersect/change-region-type "solid" (*) fluid""")
    tui.SendCommand("""/mesh/scoped-prisms/create "control-3" aspect-ratio 10 5 1.2 solid fluid-regions
 only-wall""")
    tui.SendCommand("""/mesh/auto-mesh "solid" no scoped pyramids tet""")

def ImportMesh(step):
    tui = ExtAPI.Application.ScriptByName("TUI")

    tui.SendCommand("""switch-to-meshing-mode""")
    tui.SendCommand("""/file/read-mesh "{0}" """
 .format(System.IO.Path.Combine(ExtAPI.Extension.InstallDir,"final.msh")))
    tui.SendCommand("""switch-to-solution-mode yes yes""")
    tui.SendCommand("""/display/mesh ok""")
    tui.SendCommand("""/display/views/restore-view right""")

def CreateAnalysis(step):
    tui = ExtAPI.Application.ScriptByName("TUI")

    res = System.IO.Path.GetTempFileName()
    img = System.IO.Path.GetTempFileName()+".jpg"
    curve = System.IO.Path.GetTempFileName()
    tui.SendCommand("""/define/boundary-conditions/velocity-inlet inlet no no yes yes
 no {0} no 0""".format(step.Properties["velocity"].Value), True)
    tui.SendCommand("""/solve/initialize/hyb-initialization""", True)
    tui.SendCommand("""/solve/iterate 50""", True)
    tui.SendCommand("""/display/mesh ok""", True)
    tui.SendCommand("""/display/views/restore-view right""", True)
    tui.SendCommand("""/display/set/contours/filled-contours? yes""", True)
    tui.SendCommand("""/display/set/contours/surfaces wall inlet outlet ()""", True)
    tui.SendCommand("""/display/contour pressure 0 2""", True)
    tui.SendCommand("""/display/save-picture "{0}" """.format(img), True)
    #tui.SendCommand("""/surface/delete rake-5""", True)
    tui.SendCommand("""/surface/rake-surface rake-5 0 0 0 0 0 1 50""", True)
    tui.SendCommand("""/plot/plot-direction 0 0 1""", True)
    tui.SendCommand("""/plot/plot yes "{0}" yes yes no no pressure yes 0 0 1 rake-5 ()""".format(curve), True)
    tui.SendCommand("""/report/surface-integrals/area-weighted-avg inlet () pressure
 yes "{0}" """.format(res), True)

    val = 0.
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
    n = 0;
    with System.IO.StreamReader(res) as reader:
        while n!=5:
            line = reader.ReadLine()
            if line!=None:
                n+=1
        str = line.Substring(32)
        val = System.Double.Parse(str, System.Globalization.CultureInfo.InvariantCulture)

    x=[]
    y=[]
    n = 0;
    with System.IO.StreamReader(curve) as reader:
        while n!=5:
            line = reader.ReadLine()
            if line!=None:
                n+=1
        while not line.StartsWith(")"):
            tab = line.Split("\t")
            x.Add(System.Double.Parse(tab[0], System.Globalization.CultureInfo.InvariantCulture))
            y.Add(System.Double.Parse(tab[1], System.Globalization.CultureInfo.InvariantCulture))
            line = reader.ReadLine()
    System.IO.File.Delete(res)
    System.IO.File.Delete(curve)

    step.NextStep.Properties["pressure"].Value = val
    step.NextStep.Attributes["img"] = img
    step.NextStep.Attributes["x"] = x
    step.NextStep.Attributes["y"] = y

def CreateReport(step):
    graph = step.UserInterface.GetComponent("Chart")
    graph.Title("Static Pressure")
    graph.ShowLegend(False)
    graph.Plot(step.Attributes["x"],  step.Attributes["y"], key="Static Pressure", color='g')

    help = step.UserInterface.GetComponent("Report")
    help.SetHtmlContent(ExtAPI.Extension.InstallDir,"""<center><img style="height:250px"
 src="{0}"></center>""".format(step.Attributes["img"]))
```

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

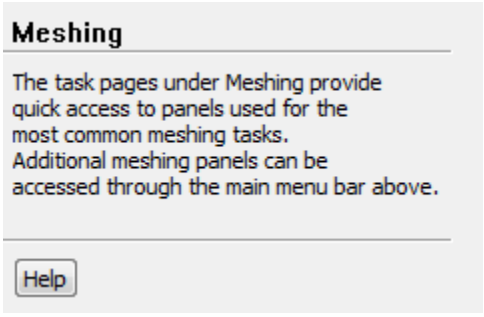
Chapter 37: Task Page Reference Guide

This reference guide provides information about the task pages in Fluent.

37.1. Meshing Task Page
37.2. Solution Setup Task Page
37.3. General Task Page
37.4. Models Task Page
37.5. Materials Task Page
37.6. Phases
37.7. Cell Zone Conditions Task Page
37.8. Boundary Conditions Task Page
37.9. Overset Interfaces Task Page
37.10. Dynamic Mesh Task Page
37.11. Reference Values Task Page
37.12. Solution Task Page
37.13. Solution Methods Task Page
37.14. Solution Controls Task Page
37.15. Solution Initialization Task Page
37.16. Calculation Activities Task Page
37.17. Run Calculation Task Page
37.18. Results Task Page
37.19. Graphics and Animations Task Page
37.20. Plots Task Page
37.21. Reports Task Page
37.22. Parameters and Customization Task Page

37.1. Meshing Task Page

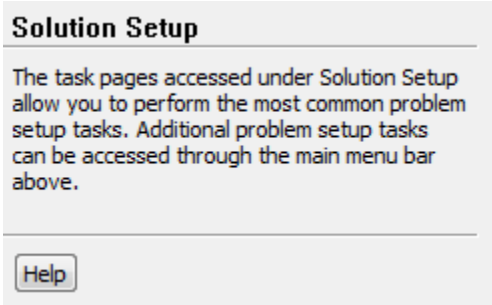
The **Meshing** task page introduces you to the most common meshing tasks involved in solving your CFD simulation using ANSYS Fluent.



37.2. Solution Setup Task Page

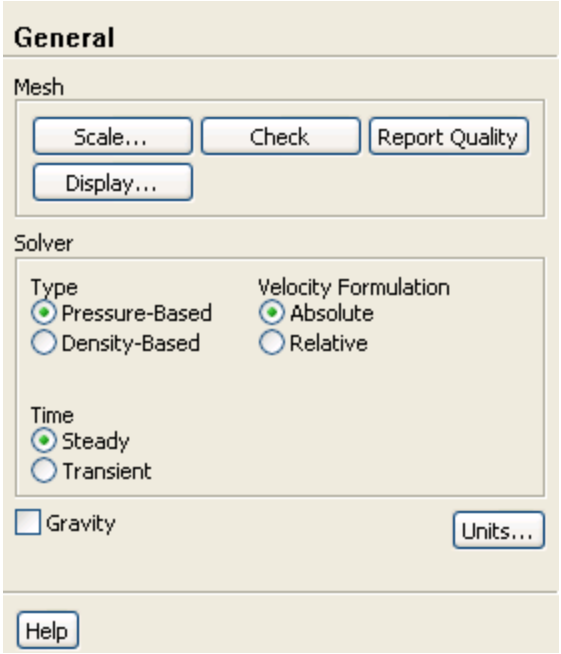
The **Solution Setup** task page introduces you to the main tasks involved in setting up your CFD simulation using ANSYS Fluent.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



37.3. General Task Page

The **General** task page allows you to set various generic problem settings, such as those related to the mesh or the solver.



Controls

Mesh

contains controls relating to mesh settings.

Scale...

opens the Scale Mesh Dialog Box (p. 2162).

Check

verifies the validity of the mesh. The check provides volume statistics, mesh topology and periodic boundary information, verification of simplex counters, and verification of node position with reference to the x axis for axisymmetric cases. See Checking the Mesh (p. 192) for details.

It is recommended that you check the mesh right after reading it into the solver, in order to detect any mesh trouble before you get started with the problem setup.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Report Quality
displays various quantities related to the quality of the mesh in the console, such as `Minimum Orthogonal Quality` and `Maximum Aspect Ratio`. See Mesh Quality (p. 131) for details.

Display...
opens the Mesh Display Dialog Box (p. 2163).

Solver
contains controls relating to solver settings.

Type
contains the solution methods available for computing a solution for your model. See Using the Solver (p. 1551) for details.

Pressure-Based
enables the pressure-based Navier-Stokes solution algorithm (the default).

Density-Based
enables the density-based Navier-Stokes coupled solution algorithm.

Velocity Formulation
specifies the velocity formulation to be used in the calculation. See Choosing the Relative or Absolute Velocity Formulation (p. 609) for details.

Absolute
enables the use of the absolute velocity formulation. This is the default setting.

Relative
enables the use of the relative velocity formulation. This option is available only with the **Pressure-Based** solver.

Time
contains options related to time dependence.

Steady
specifies that a steady flow is being solved.

Transient
enables a time-dependent solution. See Performing Time-Dependent Calculations (p. 1608) for details.

2D Space
contains options available only when solving two-dimensional problems.

Planar
indicates that the problem is two-dimensional. (This option is available only when you start the 2D version of the solver.)

Axisymmetric
indicates that the domain is axisymmetric about the x axis. When **Axisymmetric** is enabled, the 2D axisymmetric form of the governing equations is solved instead of the 2D Cartesian form. (This option is available only when you start the 2D version of the solver.) Be sure to change the zone type of the axis of rotation to **axis**, using the Boundary Conditions Task Page (p. 2389), as described in Changing Cell and Boundary Zone Types (p. 239).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Axisymmetric Swirl
specifies that the swirl component (circumferential component) of velocity is to be included in your axisymmetric model. You should select this option if you are solving swirling flow in an axisymmetric geometry (see Swirling and Rotating Flows (p. 587) for more information).

Gravity
enables the specification of gravity.

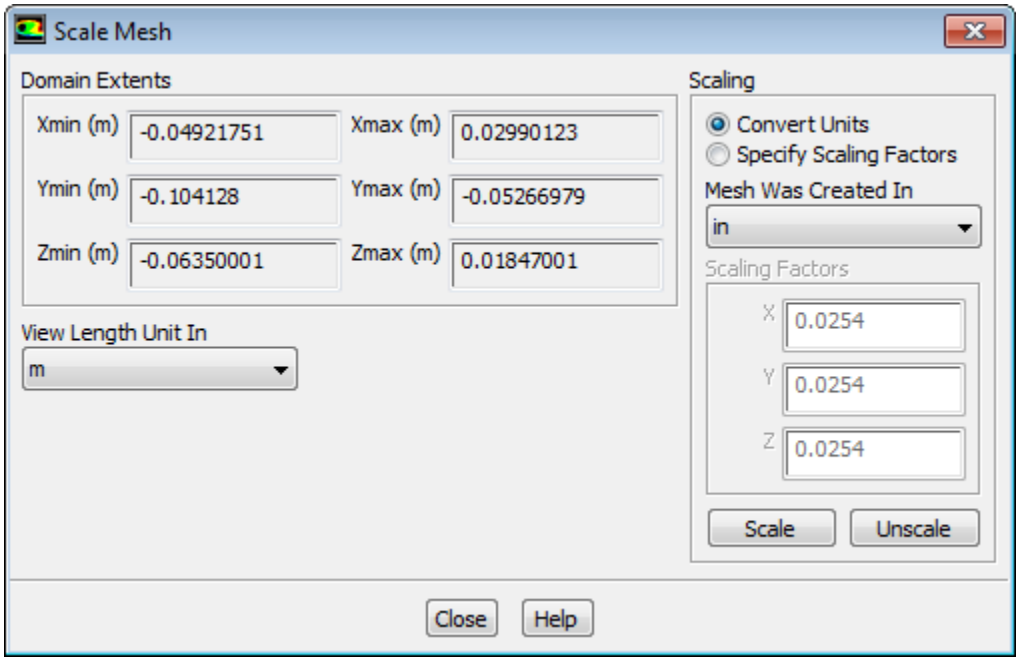
Gravitational Acceleration
sets the x, y, and z components of the gravitational acceleration vector. (The z component is available only in 3D solvers.) See Natural Convection and Buoyancy-Driven Flows (p. 833) for details about buoyancy-driven flows. This option appears only when **Gravity** is enabled.

Units...
opens the Set Units Dialog Box (p. 2166).

For additional information, see the following sections:
37.3.1. Scale Mesh Dialog Box
37.3.2. Mesh Display Dialog Box
37.3.3. Set Units Dialog Box
37.3.4. Define Unit Dialog Box
37.3.5. Mesh Colors Dialog Box

37.3.1. Scale Mesh Dialog Box

The **Scale Mesh** dialog box allows you to convert the mesh from various units of measurement to SI or to apply custom scale factors to the individual coordinates of the mesh. See Scaling the Mesh (p. 226) for details.



Controls

Domain Extents
displays the Cartesian coordinate extremes of the nodes in the mesh. (These values are not editable; they are purely informational.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Xmin, Ymin, Zmin
shows the minimum values of Cartesian coordinates in the mesh.

Xmax, Ymax, Zmax
shows the maximum values of Cartesian coordinates in the mesh.

Scaling
contains controls for converting units and setting the scale factors automatically.

Convert Units
allows you to use the conversion factors provided by ANSYS Fluent. Then indicate the units used when creating the mesh by selecting the appropriate abbreviation for meters, centimeters, millimeters, inches, or feet from the **Mesh Was Created In** drop-down list. The **Scaling Factors** will automatically be set to the correct values (for example, 0.0254 meters/inch).

Specify Scaling Factors
allows you to manually specify a scale factor in each of the Cartesian coordinate directions.

Mesh Was Created In
contains a list of common units of length. The **Scaling Factors** will automatically be set based on your selection. The units include common SI and British units such as centimeters (cm), millimeters (mm), inches (in), and feet (ft).

Scaling Factors
contains the factors applied to the mesh in each of the Cartesian coordinate directions. You can enter values manually, or use the **Mesh Was Created In** list to set scale factors automatically.

X
is the scale factor in the x direction.

Y
is the scale factor in the y direction.

Z
is the scale factor in the z direction (appears only in 3D).

Scale
multiplies each of the node coordinates by the specified scale factors.

Unscale
divides each of the node coordinates by the specified scale factors.

View Length Unit In
contains a list of common units of length. The **Domain Extents** will automatically be set based on your selection. The units include common SI and British units such as centimeters (cm), millimeters (mm), inches (in), and feet (ft). The units of length in the Set Units Dialog Box (p. 2166) will change each time you change your selection in the **View Length Unit In** drop-down list.

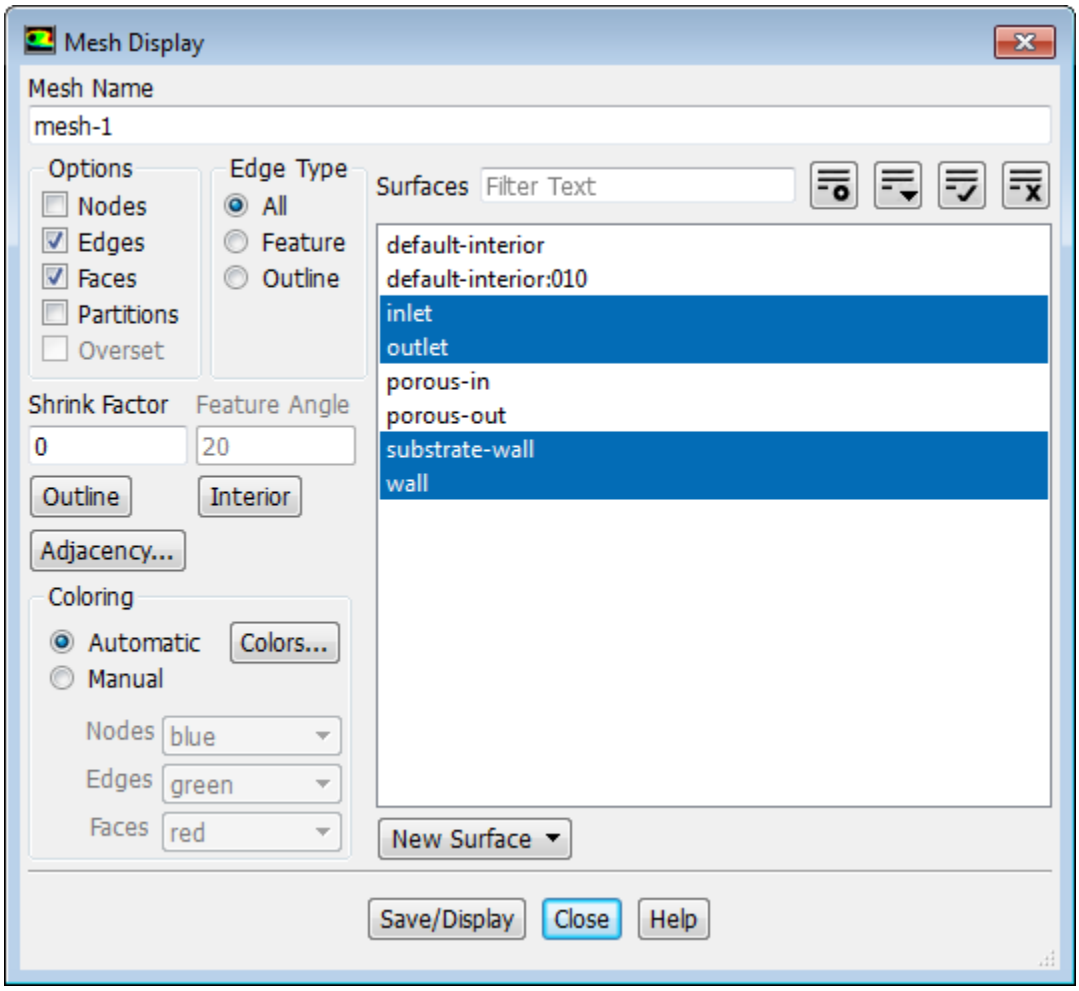
37.3.2. Mesh Display Dialog Box

The **Mesh Display** dialog box controls the display of zone, surface, and partition boundary meshes. See Displaying the Mesh (p. 1748) for details about the items below.

*You can also access the **Mesh Display** dialog box through the **Postprocessing** ribbon tab.*

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Postprocessing → Graphics → Mesh → New...



Controls

Mesh Name

is the name for a mesh display definition. You can specify a name or use the default name `mesh-id`. This control appears only for mesh display definitions.

Options

contains the rendering options described below. To see the effects of your selection you must click the **Display** button.

Nodes

enables the display of nodes on the selected surfaces.

Edges

enables the display of mesh edges on the selected surfaces.

Faces

enables the display of mesh faces (filled meshes) on the selected surfaces.

Partitions

enables the display of partition boundaries.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Overset
when enabled, displays only the solve cells for cases with overset interfaces. Receptor cells are also displayed if the text command `define/overset-interfaces/options/render-receptor-cells?` is set to `yes`.

Shrink Factor
specifies the amount to shrink faces and cells. See Shrinking Faces and Cells in the Display (p. 1754) for details.

Edge Type
controls the display of edges. (These items will not appear if the **Edges** option is turned off.)

All
enables the display of all mesh edges.

Feature
enables feature lines in an outline display. See Adding Features to an Outline Display (p. 1753) for details.

Outline
enables the display of the mesh outline.

Feature Angle
controls the amount of detail added to a feature outline display. See Adding Features to an Outline Display (p. 1753) for details. (This item will be available only if the **Feature** edge type is enabled.)

Surfaces
contains a list from which you can select the surfaces for which the mesh is to be drawn.

New Surface
is a drop-down list button that contains a list of surface options:

Point
opens the Point Surface Dialog Box (p. 2780).

Line/Rake
opens the Line/Rake Surface Dialog Box (p. 2727).

Plane
opens the Plane Surface Dialog Box (p. 2777).

Quadric
opens the Quadric Surface Dialog Box (p. 2781).

Iso-Surface
opens the Iso-Surface Dialog Box (p. 2721).

Iso-Clip
opens the Iso-Clip Dialog Box (p. 2720).

Outline
selects all "outline" boundaries in the **Surfaces** list. If all outline boundaries are already selected, it deselects them.

Interior
selects all "interior" surfaces in the **Surfaces** list. If all interior surfaces are already selected, it deselects them.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Adjacency
opens the Adjacency Dialog Box (p. 2664) from which you can identify and display face zones that are adjacent to selected cell zones.

Coloring

Automatic
nodes, edges, and faces are automatically colored by type or ID, depending on which is specified in the Mesh Colors Dialog Box (p. 2168).

Colors...
opens the Mesh Colors Dialog Box (p. 2168).

Manual
allows you to manually specify the color of nodes, edges, and faces, if enabled under **Options**.

Nodes
specify the color for the nodes.

Edges
specify the color for the edges.

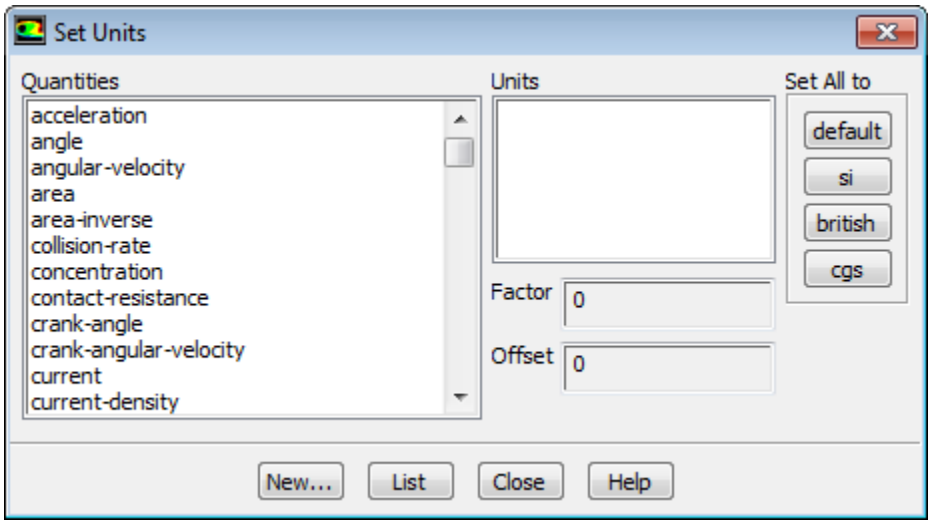
Faces
specify the color for the faces.

Display
displays the defined mesh plot.

Save/Display
displays the mesh in the active graphics window and saves the mesh display definition. This button appears only for mesh display definitions and replaces the **Display** button.

37.3.3. Set Units Dialog Box

The **Set Units** dialog box allows you to set the units system for any quantity used in ANSYS Fluent. All quantities may be set to a standard system, such as SI or British, or the units of individual quantities may be set. See Unit Systems (p. 111) for details about the items below.



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Quantities
displays the list of all quantities used by ANSYS Fluent for input and output.

Units
lists the units appropriate for the currently selected quantity. Selecting an item in the **Units** list causes that unit to be used for the currently selected quantity.

Factor
displays the conversion factor from the currently selected units to SI.

Offset
displays the conversion offset from the currently selected units to SI.

Set All to
contains standard sets of units that are applied to all quantities when selected.

default
is similar to **si**, but uses degrees instead of radians for angles.

si
selects the System International (SI) standard for all units.

british
selects the English Engineering standard for all units.

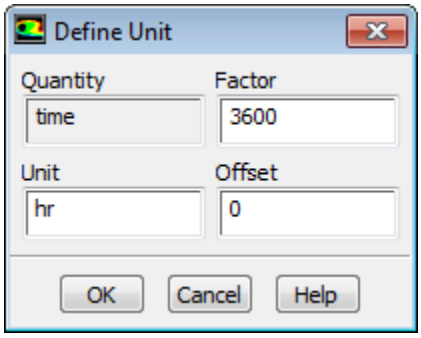
cgs
selects the CGS (centimeter-gram-second) standard for all units

New...
opens the Define Unit Dialog Box (p. 2167), in which you can specify a customized unit for a particular quantity.

List
displays the current units for all quantities in the console.

37.3.4. Define Unit Dialog Box

The **Define Unit** dialog box allows you to customize the units for a particular quantity. It is opened from the Set Units Dialog Box (p. 2166). For details about using this dialog box, see Defining a New Unit (p. 113).



Controls

Quantity
shows the name of the quantity for which you are defining a new unit. (You cannot edit this field; the quantity is selected in the Set Units Dialog Box (p. 2166).)

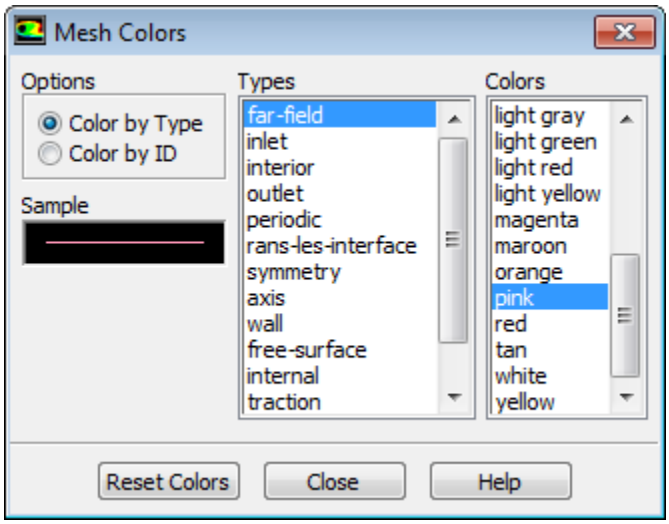
Unit
sets the name for the new unit.

Factor
sets the conversion factor from the currently selected units to SI.

Offset
sets the conversion offset from the currently selected units to SI.

37.3.5. Mesh Colors Dialog Box

The **Mesh Colors** dialog box allows you to control the colors that are used to draw meshes. It is opened from the Mesh Display Dialog Box (p. 2163). See Modifying the Mesh Colors (p. 1752) for details about the items below.



Controls

Options
contains options to select the method by which to set the colors.

Color by Type
sets the color based on the type of zone.

Color by ID
sets the color by ID.

Types
contains a selectable list of zone types. You can select the zone type for which you want to set the color.

Colors
contains a list from which you can select a color for the selected type.

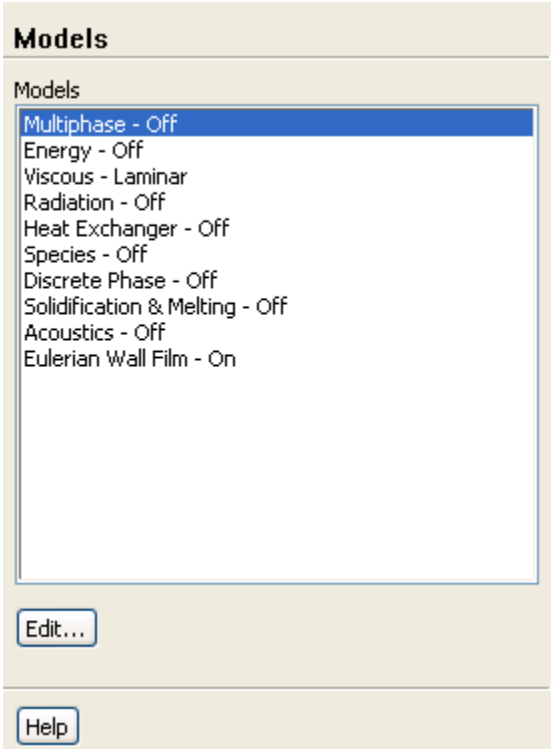
Sample
displays a sample of the currently selected color.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reset Colors
resets the mesh colors to the default selections.

37.4. Models Task Page

The **Models** task page allows you to set various generic model settings.



Controls

Models
contains a listing of the various models available in ANSYS Fluent.

You can double-click an item in the **Models** list to open the corresponding dialog box, or you can select the item in the list and click the **Edit...** button.

Multiphase
- selecting this item and clicking the **Edit...** button opens the Multiphase Model Dialog Box (p. 2171).

Energy
- selecting this item and clicking the **Edit...** button opens the Energy Dialog Box (p. 2175).

Viscous
- selecting this item and clicking the **Edit...** button opens the Viscous Model Dialog Box (p. 2176).

Radiation
- selecting this item and clicking the **Edit...** button opens the Radiation Model Dialog Box (p. 2190).

Heat Exchanger
- selecting this item and clicking the **Edit...** button opens the Heat Exchanger Model Dialog Box (p. 2200).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Species
- selecting this item and clicking the **Edit...** button opens the Species Model Dialog Box (p. 2216).

The following models are made available, depending on your setup of the **Species** dialog box.

- **Spark Ignition** - selecting this item and clicking the **Edit...** button opens the Spark Ignition Dialog Box (p. 2243). Note that spark ignition is only available for transient calculations.
- **Autoignition** - selecting this item and clicking the **Edit...** button opens the Autoignition Model Dialog Box (p. 2246). Note that autoignition is only available for transient calculations.
- **Inert** - selecting this item and clicking the **Edit...** button opens the Inert Dialog Box (p. 2249). Note that the inert model is only available when the non-premixed or partially premixed model is selected in the **Species Model** dialog box, or when a PDF file is read.
- **NOx** - selecting this item and clicking the **Edit...** button opens the NOx Model Dialog Box (p. 2250). Note that the NOx model is not compatible with premixed combustion.
- **SOx** - selecting this item and clicking the **Edit...** button opens the SOx Model Dialog Box (p. 2258). Note that the SOx model is not compatible with premixed combustion.
- **Soot** - selecting this item and clicking the **Edit...** button opens the Soot Model Dialog Box (p. 2262). Note that none of the soot models are compatible with premixed combustion.
- **Decoupled Detailed Chemistry** - selecting this item and clicking the **Edit...** button opens the Decoupled Detailed Chemistry Dialog Box (p. 2273).
- **Reacting Channel Model** - selecting this item and clicking the **Edit...** button opens the Reacting Channel Model Dialog Box (p. 2274).

Discrete Phase
- selecting this item and clicking the **Edit...** button opens the Discrete Phase Model Dialog Box (p. 2278).

Solidification & Melting
- selecting this item and clicking the **Edit...** button opens the Solidification and Melting Dialog Box (p. 2288).

Acoustics
- selecting this item and clicking the **Edit...** button opens the Acoustics Model Dialog Box (p. 2289).

Eulerian Wall Film
- selecting this item and clicking the **Edit...** button opens the Eulerian Wall Film Dialog Box (p. 2294).

Edit...
displays the dialog box corresponding to the selected item in the **Models** list.

For additional information, see the following sections:
37.4.1. Multiphase Model Dialog Box
37.4.2. Energy Dialog Box
37.4.3. Viscous Model Dialog Box
37.4.4. Radiation Model Dialog Box
37.4.5. View Factors and Clustering Dialog Box
37.4.6. Participating Boundary Zones Dialog Box
37.4.7. Solar Calculator Dialog Box
37.4.8. Heat Exchanger Model Dialog Box

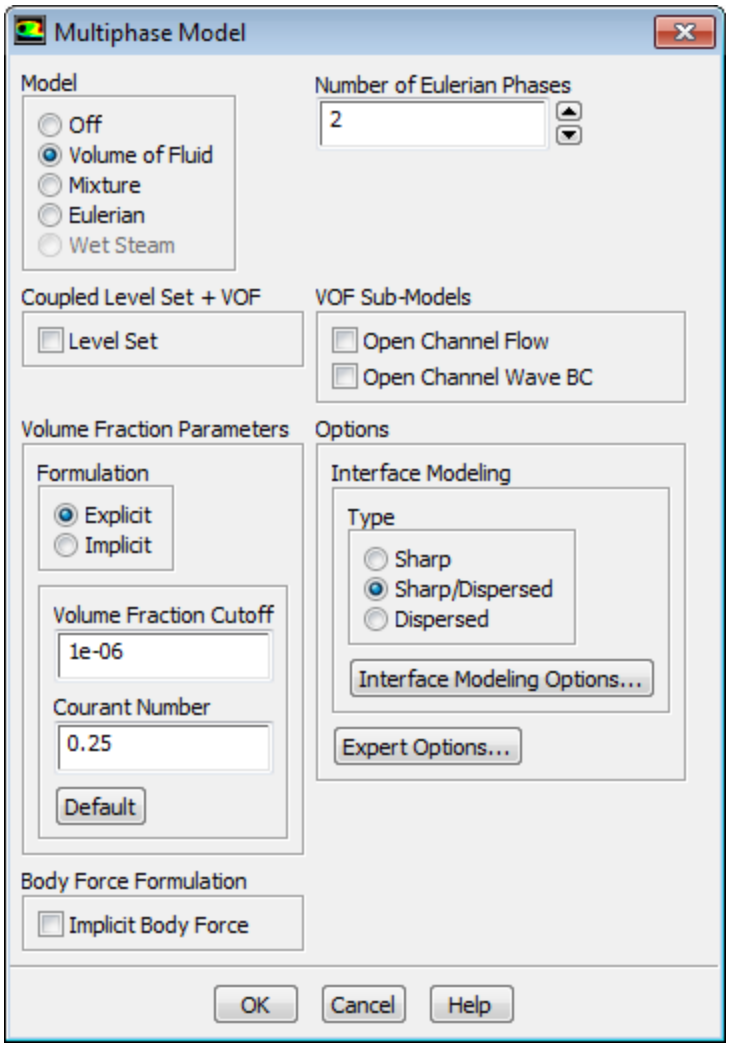
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.4.9. Dual Cell Heat Exchanger Dialog Box
37.4.10. Set Dual Cell Heat Exchanger Dialog Box
37.4.11. Heat Transfer Data Table Dialog Box
37.4.12. NTU Table Dialog Box
37.4.13. Copy From Dialog Box
37.4.14. Ungrouped Macro Heat Exchanger Dialog Box
37.4.15. Velocity Effectiveness Curve Dialog Box
37.4.16. Core Porosity Model Dialog Box
37.4.17. Macro Heat Exchanger Group Dialog Box
37.4.18. Species Model Dialog Box
37.4.19. Coal Calculator Dialog Box
37.4.20. Integration Parameters Dialog Box
37.4.21. Flamelet 3D Surfaces Dialog Box
37.4.22. Flamelet 2D Curves Dialog Box
37.4.23. Unsteady Flamelet Parameters Dialog Box
37.4.24. Flamelet Fluid Zones Dialog Box
37.4.25. PDF Table Dialog Box
37.4.26. Spark Ignition Dialog Box
37.4.27. Set Spark Ignition Dialog Box
37.4.28. Autoignition Model Dialog Box
37.4.29. Inert Dialog Box
37.4.30. NOx Model Dialog Box
37.4.31. SOx Model Dialog Box
37.4.32. Soot Model Dialog Box
37.4.33. Sticking Coefficients Dialog Box
37.4.34. Mechanism Dialog Box
37.4.35. Reactor Network Dialog Box
37.4.36. Decoupled Detailed Chemistry Dialog Box
37.4.37. Reacting Channel Model Dialog Box
37.4.38. Reacting Channel 2D Curves Dialog Box
37.4.39. Discrete Phase Model Dialog Box
37.4.40. DEM Collisions Dialog Box
37.4.41. Create Collision Partner Dialog Box
37.4.42. Copy Collision Partner Dialog Box
37.4.43. Rename Collision Partner Dialog Box
37.4.44. DEM Collision Settings Dialog Box
37.4.45. Solidification and Melting Dialog Box
37.4.46. Acoustics Model Dialog Box
37.4.47. Acoustic Sources Dialog Box
37.4.48. Acoustic Receivers Dialog Box
37.4.49. Interior Cell Zone Selection Dialog Box
37.4.50. Eulerian Wall Film Dialog Box
37.4.51. Potential Dialog Box

37.4.1. Multiphase Model Dialog Box

The **Multiphase Model** dialog box allows you to set parameters for modeling multiphase flow. See Enabling the Multiphase Model (p. 1367) – Including Cavitation Effects (p. 1455) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Model

allows you to select one of four multiphase models.

Off

disables the calculation of multiphase flow.

Volume of Fluid

enables the VOF model described in Volume of Fluid (VOF) Model Theory in the Theory Guide. See Setting Up the VOF Model (p. 1408) for details about using the model. This is available only with the pressure-based solver.

Mixture

enables the mixture model described in Mixture Model Theory in the Theory Guide. See Setting Up the Mixture Model (p. 1447) for details about using the model. This is available only with the pressure-based solver.

Eulerian

enables the Eulerian model described in Eulerian Model Theory in the Theory Guide. See Setting Up the Eulerian Model (p. 1455) for details about using the model. This is available only with the pressure-based solver.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Wet Steam
enables the wet steam model described in Wet Steam Model Theory in the Theory Guide. See Setting Up the Wet Steam Model (p. 1495) for details about using the model. This is available only with the density-based solver.

Number of Eulerian Phases
allows you to specify the number of phases for the multiphase calculation. You can specify up to 20 phases.

Coupled Level Set + VOF
allows you to apply an interface tracking method that couples the level set method with the VOF formulation.

Volume Fraction Parameters
contains parameters related to the VOF and Eulerian model. This section of the dialog box will appear only when **Volume of Fluid** or **Eulerian** is the selected **Model**.

Formulation
allows you to select the desired volume tracking formulation.

Explicit
enables the Euler explicit formulation, described in The Explicit Formulation in the Theory Guide. See Choosing Volume Fraction Formulation (p. 1371) for more information.

Implicit
enables the implicit formulation, described in The Implicit Formulation in the Theory Guide. See Choosing Volume Fraction Formulation (p. 1371) for more information.

Volume Fraction Cutoff
specifies a cutoff limit for the volume fraction values. The value that you provide is used as the lower cutoff for the volume fraction. All volume fraction values in the domain below this cutoff value are set to zero. The upper cutoff is calculated as (1.0 - lower cutoff). All volume fraction values above the upper cutoff value are set to 1.0. The default value is 1e-6.

Important

The **Volume Fraction Cutoff** value can be specified when using the VOF model, or when using the Eulerian Multiphase model with the **Explicit** scheme.

Courant Number
specifies the maximum Courant number allowed near the free surface. This item will not appear if the **Implicit Scheme** is selected. See Setting Time-Dependent Parameters for the Explicit Volume Fraction Formulation (p. 1445) for details.

Mixture Parameters
contains options related to the **Mixture** model.

Slip Velocity
enables/disables the calculation of slip velocities for the secondary phases as described in Relative (Slip) Velocity and the Drift Velocity in the Theory Guide. See also Solving a Homogeneous Multiphase Flow (p. 1375).

Eulerian Parameters
contains options related to the **Eulerian** model.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Dense Discrete Phase Model
allows you to include the Dense Discrete Phase model (see Including the Dense Discrete Phase Model (p. 1481) for details). Enabling this model automatically enables the DPM model.

Multi-Fluid VOF Model
allows you to include the multi-fluid VOF model (see Including the Multi-Fluid VOF Model (p. 1493) for details). The multi-fluid VOF model allows the modeling of interface sharpening schemes and free surface flow.

Boiling Model
allows you to include the Boiling model (see Including the Boiling Model (p. 1486) for details).

Boiling Model Options
allows you to select the type of boiling model to apply to your case.

RPI Boiling Model
is where the total heat flux from the wall to the liquid is partitioned into three components, namely the convective heat flux, the quenching heat flux, and the evaporative heat flux. Details about this model can be found in RPI Model in the Theory Guide.

Non-equilibrium Boiling
is a modification to the RPI model in order to model different boiling regimes like DNB and critical heat flux. Details about this model can be found in Non-equilibrium Subcooled Boiling in the Theory Guide.

Critical Heat Flux
is where the critical heat flux condition is characterized by a sharp reduction of local heat transfer coefficients and the excursion of wall surface temperatures. Details about this model can be found in Critical Heat Flux in the Theory Guide.

Evaporation-Condensation
allows you to select the evaporation-condensation model to use in your simulation.

Evaporation-Condensation Model Options

Lee Model
enables the Lee Model, which is the same model that is used with VOF and the mixture model. Details about this model can be found in Lee Model in the *Fluent Theory Guide*.

Thermal Phase Change
enables the thermal phase change model. Details about this model can be found in Thermal Phase Change Model in the *Fluent Theory Guide*.

Number of Discrete Phase
allows you to specify the number of discrete phases when the **Dense Discrete Phase Model** option is enabled.

Body Force Formulation
contains an additional option for body force calculations.

Implicit Body Force
enables the implicit body force treatment described in Including Body Forces (p. 1377).

VOF Sub-Models
contains additional sub-models that can be applied when using the VOF model.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Open Channel Flow
enables the model to study the effects of open channel flow. See Open Channel Flow in the Theory Guide and Modeling Open Channel Flows (p. 1410) for details.

Open Channel Wave BC
enables the model to set specific parameters for a particular boundary for open channel wave boundaries. See Open Channel Wave Boundary Conditions in the Theory Guide and Modeling Open Channel Wave Boundary Conditions (p. 1419) for details.

Options
contains additional modeling options.

Interface Modeling
contains settings for how to model interfaces for the VOF model and Eulerian with Multi-Fluid VOF enabled. See Interface Modeling Type (p. 1371) for details.

Type
allows you to select the type of interface modeling to be performed. You can choose between **Sharp**, **Dispersed**, or a hybrid **Sharp/Dispersed** when both types of interfaces may be present, or when the interface is mildly sharp. See Interface Modeling Type (p. 1371) for more details.

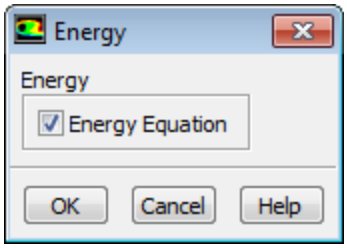
Interfacial Anti-Diffusion
enables an anti-diffusion treatment at the interface when using the **Sharp** interface modeling type. This can be used to reduce the effects of numerical diffusion that may occur with coarse meshes, high-aspect ratio cells, or large jumps in cell volume near the interface.

Interface Modeling Options...
opens the **Interface Modeling Options** dialog box from which you can enable **Zonal Discretization** and/or **Phase Localized Discretization**.

Expert Options...
opens the **Expert Options** dialog box which contains settings for VOF time advancement when using the Explicit formulation. Here you can set the **Sub-Time Step Calculation Method** and enable **Solve VOF Every Iteration**. See Expert Options (p. 1374) for details.

37.4.2. Energy Dialog Box

The **Energy** dialog box allows you to set parameters related to energy or heat transfer in your model.



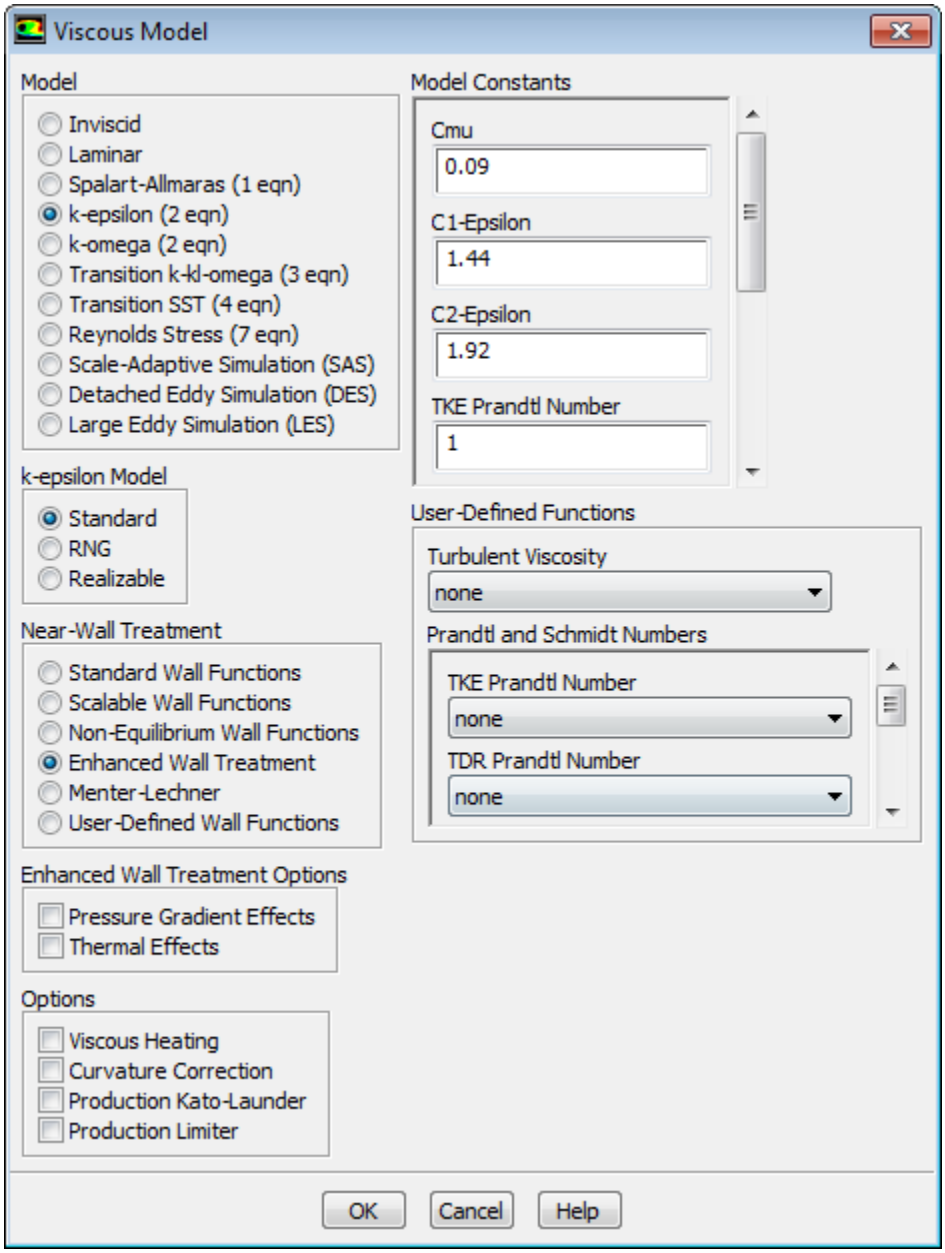
Controls

Energy
contains inputs related to the modeling of energy.

Energy Equation
enables/disables the calculation of energy in the model.

37.4.3. Viscous Model Dialog Box

The **Viscous Model** dialog box allows you to set parameters for inviscid, laminar, and turbulent flow. See Steps in Using a Turbulence Model (p. 764) for details about using this dialog box to set up a turbulent flow calculation.



Controls

Model

contains options for specifying the viscous model.

Inviscid

specifies inviscid flow.

Laminar

specifies laminar flow.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Spalart-Allmaras
specifies turbulent flow to be calculated using the Spalart-Allmaras model. (See Spalart-Allmaras Model in the Theory Guide for background about this model. See Setting Up the Spalart-Allmaras Model (p. 766) for details about using this model.)

k-epsilon
specifies turbulent flow to be calculated using one of three k-ε models. (See Standard, RNG, and Realizable k-ε Models in the Theory Guide for background about this model. See Setting Up the k-ε Model (p. 767) for details about using this model.)

k-omega
specifies turbulent flow to be calculated using one of two k-ω models. (See Standard, BSL, and SST k-ω Models in the Theory Guide for background about these models. See Setting Up the k-ω Model (p. 771) for details about using this model.)

Transition k-kl-omega
specifies turbulent flow to be calculated using the Transition k-kl-ω model. (See k-kl-ω Transition Model in the Theory Guide for background about this model. See Setting Up the Transition k-kl-ω Model (p. 775) for details about using this model.)

Transition SST
specifies turbulent flow to be calculated using the Transition SST model. (See Transition SST Model in the Theory Guide for background about this model. See Setting Up the Transition SST Model (p. 775) for details about using this model.)

Reynolds Stress
specifies turbulent flow to be calculated using the RSM. (See Reynolds Stress Model (RSM) in the Theory Guide for background about this model. See Setting Up the Reynolds Stress Model (p. 778) for details about using this model.)

Scale-Adaptive Simulation (SAS)
specifies turbulent flow to be calculated using the SAS model in combination with the SST k-ω model. (See Scale-Adaptive Simulation (SAS) Model in the Theory Guide for background about this model. See Setting Up Scale-Adaptive Simulation (SAS) Modeling (p. 781) for details about using this model.)

Detached Eddy Simulation
specifies turbulent flow to be calculated using the DES model. (See Detached Eddy Simulation (DES) in the Theory Guide for background about this model. See Setting Up the Detached Eddy Simulation Model (p. 783) for details about using this model.)

Large Eddy Simulation
(3D only) specifies turbulent flow to be calculated using the LES model. (See Large Eddy Simulation (LES) Model in the Theory Guide for background about this model. See Setting Up the Large Eddy Simulation Model (p. 792) for details about using this model.)

Spalart-Allmaras Production
contains options for the Spalart-Allmaras model. This portion of the dialog box will appear only if **Spalart-Allmaras** is selected as the **Model**.

Vorticity-Based
selects the vorticity-based calculation of the deformation tensor S (see Equation 4.22 in the Theory Guide).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Strain/Vorticity-Based
selects the strain/vorticity-based calculation of the deformation tensor S (see Equation 4.24 in the Theory Guide).

k-epsilon Model
contains options for specifying which of the k-ε models is to be used. This portion of the dialog box will appear only if **k-epsilon** is selected as the **Model**.

Standard
selects the standard k-ε model, described in Standard k-ε Model in the Theory Guide and Setting Up the k-ε Model (p. 767).

RNG
selects the RNG k-ε model, described in RNG k-ε Model. in the Theory Guide and Setting Up the k-ε Model (p. 767).

Realizable
selects the realizable k-ε model, described in Realizable k-ε Model in the Theory Guide and Setting Up the k-ε Model (p. 767).

RNG Options
specifies parameters that affect the solution of problems solved with the RNG k-ε model. This portion of the dialog box will appear only if **RNG** is selected as the **k-epsilon Model**.

Differential Viscosity Model
specifies whether or not the low-Reynolds-number RNG modifications to turbulent viscosity should be included. By default, this option is turned off. It is likely to have an effect only when the near-wall regions in the domain are well resolved in terms of mesh density. See Differential Viscosity Modification (p. 800) for details.

Swirl Dominated Flow
specifies whether or not the RNG modification to turbulent viscosity for swirling flows should be included. This option is available only in 3D and 2D axisymmetric swirl solvers, and it can yield improved predictions when solving flows with significant swirl. See Swirl Modification (p. 801) for details.

k-omega Model
contains options for specifying which of the k-ω models is to be used. This portion of the dialog box will appear only if **k-omega** is selected as the **Model**.

Standard
selects the standard k-ω model, described in Standard k-ω Model in the Theory Guide and Setting Up the k-ω Model (p. 771).

BSL
selects the baseline (BSL) k-ω model, described in Baseline (BSL) k-ω Model in the Theory Guide and Setting Up the k-ω Model (p. 771).

SST
selects the shear-stress transport (SST) k-ε model, described in Shear-Stress Transport (SST) k-ω Model in the Theory Guide and Setting Up the k-ω Model (p. 771).

k-omega Options
specifies parameters that affect the solution of problems solved with the k-ω models. This portion of the dialog box will appear only if **k-omega** is selected as the **Model**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Low-Re Corrections
specifies whether corrections that improve the accuracy in predicting low Reynolds number flows should be included. This option is available only for the k-ω models and the stress-omega RSM model. See Low-Re Corrections (p. 801) for details.

Shear Flow Corrections
specifies whether corrections that improve the accuracy in predicting free shear flows should be included. This option is available only for the standard k-ω model and the stress-omega RSM model. See Shear Flow Corrections (p. 801) for details.

Turbulence Damping
is required for accurate modeling of the interfacial area. This option is available for the standard, BSL, and SST k-ω models Enter the desired **Damping Factor**, which by default is set to 10. See Turbulence Damping (p. 801) for details.

Transition SST Options
allows you to include the **Roughness Correlation** of rough walls as described in Transition SST and Rough Walls in the Theory Guide.

Roughness Correlation
when enabled allows you to specify the **Geometric Roughness Height** as a constant value.

Reynolds-Stress Model
specifies the various Reynolds stress models (RSM).

Linear Pressure-Strain
enables the linear pressure-strain model. See Linear Pressure-Strain Model in the Theory Guide for details.

Quadratic Pressure-Strain
enables the quadratic pressure-strain model for superior performance in a range of basic shear flows, including plane strain, rotating plane shear, and axisymmetric expansion/contraction. See Quadratic Pressure-Strain Model in the Theory Guide for details. Note that this option cannot be used with the **Wall Reflection Effects** option or the **Enhanced Wall Treatment**.

Stress-Omega
enables a stress-transport model that is based on the omega equations and LRR model [130] (p. 2894). This model is ideal for modeling flows over curved surfaces and swirling flows. See Stress-Omega Model in the Theory Guide for details.

Stress-BSL
enables a linear model for the pressure-strain term like the **Stress-Omega** model, but solves the scale equation from the baseline (BSL) k-ω model, and thus removes the free-stream sensitivity observed with the **Stress-Omega** model. Like the **Stress-Omega** model, it is recommended for modeling flows over curved surfaces and swirling flows. See Stress-BSL Model in the Theory Guide for details.

Reynolds-Stress Options
specifies parameters that affect the solution of problems solved with the Reynolds stress model. This portion of the dialog box will appear only if **Reynolds Stress** is selected as the **Model**.

Wall BC from k Equation
enables the explicit setting of boundary conditions for the Reynolds stresses near the walls, using the values computed with Equation 4.238 in the Theory Guide. See Solving the k Equation to Obtain Wall Boundary Conditions (p. 802) for details. This option is on by default.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Wall Reflection Effects
enables the calculation of the component of the pressure strain term responsible for the redistribution of normal stresses near the wall. See Including the Wall Reflection Term (p. 802) for details. Note that this option is not available if you have enabled the **Quadratic Pressure-Strain Model**.

RANS Model
contains options for the subgrid-scale model used by the **Detached Eddy Simulation Model**. This portion of the dialog box will appear only if **Detached Eddy Simulation Model** is selected as the **Model**.

Spalart-Allmaras
enables the Spalart-Allmaras RANS model. See Detached Eddy Simulation (DES) in the Theory Guide for details.

Realizable k-epsilon
enables the Realizable k-ε RANS model. See Detached Eddy Simulation (DES) in the Theory Guide for details.

SST k-omega
enables the SST k-ω RANS Model. See Detached Eddy Simulation (DES) in the Theory Guide for details.

DES Options
contain the option to include a delayed Detached Eddy Simulation.

Delayed DES
is useful for RANS meshes with high aspect ratios in the boundary layer. This option preserves the RANS model throughout the boundary layer. (See Delayed Detached Eddy Simulation (DDES) (p. 800) for details.)

Subgrid-Scale Model
contains options for the subgrid-scale model used by the LES model. This portion of the dialog box will appear only if **Large Eddy Simulation** is selected as the **Model**.

Smagorinsky-Lilly
selects the Smagorinsky-Lilly subgrid-scale model described in Subgrid-Scale Models in the Theory Guide.

WALE
selects the Wall-Adapting local Eddy-Viscosity model described in Wall-Adapting Local Eddy-Viscosity (WALE) Model in the Theory Guide.

WMLES
selects the algebraic Wall-Modeled LES model described in Algebraic Wall-Modeled LES Model (WMLES) in the Theory Guide.

WMLES S-Omega
selects the algebraic Wall-Modeled LES S-Ω model described in Algebraic WMLES S-Omega Model Formulation in the Theory Guide.

Kinetic-Energy Transport
selects the dynamic kinetic energy subgrid-scale model described in Dynamic Kinetic Energy Subgrid-Scale Model in the Theory Guide.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LES Model Options

contains options for the Large Eddy Simulation model. This portion of the dialog box will appear only if **Large Eddy Simulation** is selected as the **Model**.

Dynamic Stress

enables the dynamic stress model. It is available for selection when the **Smagorinsky-Lilly** is selected, and is enabled (and cannot be disabled) when the **Kinetic-Energy Transport** is selected.

Dynamic Energy Flux

enables the dynamic energy flux model. It is available when the LES model option **Dynamic Stress** is enabled.

Dynamic Scalar Flux

enables the dynamic computation of turbulent Sc (σ_t in Equation 8.5 in the Theory Guide). See Definition of the Mixture Fraction in the Theory Guide for details. It is available when the LES model option **Dynamic Stress** is enabled.

Dynamic Fvar

enables the dynamic mixture fraction variance model. It is available when **Non-Premixed Combustion** or **Partially Premixed Combustion** is selected in the Species Model Dialog Box (p. 2216). See The Non-Premixed Model for LES in the Theory Guide for details.

Near-Wall Treatment

specifies the near-wall treatment to be used for modeling turbulence. See Near-Wall Treatments for Wall-Bounded Turbulent Flows in the Theory Guide for details about the available methods. This portion of the dialog box will appear if **k-epsilon** or **Reynolds Stress** is selected as the **Model**.

Standard Wall Functions

enables the use of standard wall functions (described in Standard Wall Functions in the Theory Guide).

Scalable Wall Functions

enables the use of scalable wall functions (described in Scalable Wall Functions in the Theory Guide).

Non-Equilibrium Wall Functions

enables the use of non-equilibrium wall functions (described in Non-Equilibrium Wall Functions in the Theory Guide).

Enhanced Wall Treatment

enables the use of the enhanced wall treatment (described in Enhanced Wall Treatment ε-Equation (EWT-ε) in the Theory Guide). Note that this option will not appear if you have enabled the **Quadratic Pressure-Strain Model** under **Reynolds-Stress Options**.

Menter-Lechner

enables the use of the Menter-Lechner near-wall treatment (described in Menter-Lechner ε-Equation (ML-ε) in the Theory Guide). Note that this option is only available when **k-epsilon** is selected for the **Model**.

User-Defined Wall Functions

enables you to hook a user-defined function, used to define the **Law of the Wall**. See User-Defined Wall Functions in the Theory Guide for more information.

Enhanced Wall Treatment Options

allows you to include pressure gradient or thermal effects in the calculation. See Near-Wall Treatments for Wall-Bounded Turbulent Flows in the Theory Guide.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pressure Gradient Effects
enables the effect of pressure gradient.

Thermal Effects
enables thermal effects in the calculation. This option appears only if the energy equation is enabled.

Options
contains general options for viscous modeling.

Viscous Heating
(if enabled) includes the viscous dissipation terms in the energy equation. This option is recommended when you are solving a compressible flow. Note that this option is always turned on when one of the density-based solvers is used; you will not be able to turn it off.

Low-Pressure Boundary Slip
includes slip boundary conditions for velocity and temperature for modeling fluid flow at very low pressures as in semiconductor fabrication devices. See Slip Boundary Formulation for Low-Pressure Gas Systems in the Theory Guide. This option is available only for laminar flows.

Full Buoyancy Effects
enables the inclusion of buoyancy effects on ε. See Including Turbulence Generation Due to Buoyancy (p. 798) for details. This option will appear if **k-epsilon** or **Reynolds Stress** is selected as the **Model** *and* a nonzero gravitational acceleration has been specified in the Operating Conditions Dialog Box (p. 2382).

Curvature Correction
when enabled, modifies the turbulence production term to sensitize the standard eddy-viscosity models to the effects of streamline curvature and system rotation. This is available for the **Spalart-Allmaras**, **k-epsilon**, **k-omega**, **Transition SST**, **Scale-Adaptive Simulation**, and **Detached Eddy Simulation** with the **SST k-omega** model.

Compressibility Effects
when enabled, can improve the prediction of free shear layers at high Mach numbers. This is available when the compressible form of the ideal gas law or the real-gas model is activated for **k-epsilon**, **k-omega**, **Transition k-kl-omega**, **Reynolds Stress**, **Scale-Adaptive Simulation**, and **Detached Eddy Simulation** with the **Realizable k-epsilon** model. For details, see Model Enhancements (p. 755).

Mixture Drift Force
includes the effect of turbulent drift velocity when using the **Mixture** model. See Modeling Turbulence (p. 1474). Note that this option is only available when using the **Mixture** model with **Slip Velocity** enabled in the **Multiphase Model** dialog box. See Including Mixture Drift Force (p. 1454).

Production Kato-Launder
when enabled, the Kato-Launder modification for the production term limits the production term in the turbulence equation. For details see, Production Limiters for Two-Equation Models in the *Fluent Theory Guide*.

Production Limiter
when enabled, limits the production term in the turbulence equation. For details, see Production Limiters for Two-Equation Models in the *Fluent Theory Guide*.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Intermittency Transition Model
enables the Intermittency Transition model to account for transitional effects. This option is only available for the BSL k-ω, SST k-ω, Scale-Adaptive Simulation with BSL / SST, and Detached Eddy Simulation with BSL / SST models. For details, see Intermittency Transition Model in the Theory Guide.

Intermittency Transition Options
contains an option for the Intermittency Transition model. This group box is only available when the **Intermittency Transition Model** option is enabled.

Include Crossflow Transition
includes the effects of crossflow instability. For details, see Transport Equations for the Intermittency Transition Model in the Theory Guide.

Turbulence Multiphase Model
contains options for multiphase turbulence models. This portion of the dialog box will appear if **Eulerian** is selected as the **Model** in the Multiphase Model Dialog Box (p. 2171).

Mixture
specifies the (default) mixture turbulence model.

Dispersed
specifies the dispersed turbulence model.

Per Phase
specifies the calculation of a set of turbulence equations for each phase.

See Turbulence Models in the Theory Guide for details about the available multiphase turbulence models.

Model Constants
contains constants used in the equations for turbulence. See Spalart-Allmaras Model, Standard k-ε Model, RNG k-ε Model, k-kl-ω Transition Model, Transition SST Model, Realizable k-ε Model, Reynolds Stress Model (RSM), Standard k-ω Model, Shear-Stress Transport (SST) k-ω Model, and Large Eddy Simulation (LES) Model in the Theory Guide for details about these constants.

Cb1
(only for the Spalart-Allmaras model) is the constant C_{b1} in Equation 4.19 in the Theory Guide.

Cb2
(only for the Spalart-Allmaras model) is the constant C_{b2} in Equation 4.15 in the Theory Guide.

Cv1
(only for the Spalart-Allmaras model) is the constant C_{v1} in Equation 4.17 in the Theory Guide.

Cw2
(only for the Spalart-Allmaras model) is the constant C_{w2} in Equation 4.28 in the Theory Guide.

Cw3
(only for the Spalart-Allmaras model) is the constant C_{w3} in Equation 4.27 in the Theory Guide.

Cprod
(only for the Spalart-Allmaras model when the **Strain/Vorticity-Based Production** option is used) is the constant C_{prod} in Equation 4.24 in the Theory Guide.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cmu
(only for the standard or RNG k-ε model, the RSM, or the k-kl-ω Transition model) is the constant C_μ that is used to compute μ_t.

C1-Epsilon
(only for the standard or RNG k-ε model or the RSM) is the constant $C_{1\varepsilon}$ used in the transport equation for ε.

C2-Epsilon
(only for the standard, RNG, or realizable k-ε model or the RSM) is the constant $C_{2\varepsilon}$ used in the transport equation for ε.

C3-Epsilon
(only for the dispersed or per-phase k-ε multiphase models) is the constant $C_{3\varepsilon}$ in Equation 17.358 in the Theory Guide.

C-lambda
(only for the k-kl-ω Transition model) is the constant C_λ in the definition of the effective length,

CR
(only for the k-kl-ω Transition model) is the constant C_R used in the definition of R, where R represents the averaged effect of the breakdown of streamwise fluctuations into turbulence during bypass transition

ANAT
(only for the k-kl-ω Transition model) is the constant A_{NAT}

ATS
(only for the k-kl-ω Transition model) is the constant A_{TS}

CNAT, crit
(only for the k-kl-ω Transition model) is the constant $C_{NAT,crit}$

CTS, crit
(only for the k-kl-ω Transition model) is the constant $C_{TS,crit}$

CRNAT
(only for the k-kl-ω Transition model) is the constant $C_{R,NAT}$

Anu
(only for the k-kl-ω Transition model) is the constant A_ν

CINT
(only for the k-kl-ω Transition model) is the constant C_{INT}

Cw1
(only for the k-kl-ω Transition model) is the constant $C_{\omega 1}$

Cw3
(only for the k-kl-ω Transition model) is the constant $C_{\omega 3}$

Calpha-teta
(only for the k-kl-ω Transition model) is the constant $C_{\alpha,\theta}$

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ctaul
(only for the k-kl-ω Transition model) is the constant $C_{\tau,1}$

SDR Prandtl Number
(only for the standard k-ω model and the k-kl-ω Transition model) is the effective "Prandtl" number for the transport of the specific dissipation rate, σ_ω.

Ca1
(only for the Transition SST model)

Ca2
(only for the Transition SST model)

Ce1
(only for the Transition SST model)

Ce2
(only for the Transition SST model)

C_thetat
(only for the Transition SST model)

C_s1
(only for the Transition SST model)

Intermit. Prandtl #)
(only for the Transition SST model)

Re_theta. Prandtl #)
(only for the Transition SST model)

C Shielded DES
(only for the SDES or SBES model) is a constant that serves the same purpose as C_{d1} in the DDES turbulence model (see Equation 4.262 in the Theory Guide).

Csdes
(only for the SDES or SBES model) is C_{SDES}, as described in Shielded Detached Eddy Simulation (SDES) in the Theory Guide.

Cwale
(only for the WALE subgrid-scale model of the LES or SBES model) is C_w, as described in Wall-Adapting Local Eddy-Viscosity (WALE) Model in the Theory Guide.

Swirl Factor
sets the value of α_s in Equation 4.46 in the Theory Guide. This item appears for the RNG k-ε model when the **Swirl Dominated Flow** option is turned on.

Alpha*_inf
(only for the standard, BSL, or SST k-ω model, and the Transition SST model) is the constant α^*_∞ in Equation 4.74 in the Theory Guide.

Alpha_inf
(only for the standard, BSL, or SST k-ω model, and the Transition SST model) is the constant α_∞ in Equation 4.82 in the Theory Guide.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Alpha_0
(only for the standard, BSL, or SST k-ω model with the **Low-Re Corrections** option enabled) is the constant α_0 in Equation 4.82 in the Theory Guide.

Beta*_inf
(only for the standard, BSL, or SST k-ω model, and the Transition SST model) is the constant β^*_∞ in Equation 4.87 in the Theory Guide.

Beta_i
(only for the standard k-ω model) is the constant β_i in Equation 4.95 in the Theory Guide.

R_beta
(only for the standard, BSL, or SST k-ω model) is the constant R_β in Equation 4.87 in the Theory Guide.

R_k
(only for the standard, BSL, or SST k-ω model with the **Low-Re Corrections** option enabled) is the constant R_k in Equation 4.74 in the Theory Guide.

R_w
(only for the standard, BSL, or SST k-ω model with the **Low-Re Corrections** option enabled) is the constant R_ω in Equation 4.82 in the Theory Guide.

Zeta*
(only for the standard, BSL, or SST k-ω model when the **Compressibility Effects** option is enabled) is the constant ζ^* in Equation 4.86 in the Theory Guide.

Mt0
(only for the standard, BSL, or SST k-ω model when the **Compressibility Effects** option is enabled) is the constant M_{t0} in Equation 4.96 in the Theory Guide.

a1
(only for the SST k-ω model, and the Transition SST model) is the constant a_1 in Equation 4.117 in the Theory Guide.

Beta_i (Inner)
(only for the BSL or SST k-ω model, and the Transition SST model) is the constant $\beta_{i,1}$ in Model Constants in the Theory Guide.

Beta_i (Outer)
(only for the BSL or SST k-ω model, and the Transition SST model) is the constant $\beta_{i,2}$ in Model Constants in the Theory Guide.

Cs
is a constant that depends on the turbulence model. For LES, it is the Smagorinsky constant C_s in Equation 4.285 in the Theory Guide. For SAS, it is the high wave number damping coefficient C_S in Equation 4.248 in the Theory Guide.

C1-PS
(only for RSM) is the constant C_1 in Equation 4.209 in the Theory Guide.

C2-PS
(only for RSM) is the constant C_2 in Equation 4.210 in the Theory Guide.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

C1′-PS

(only for RSM) is the constant $C_1^{'}$ in Equation 4.211 in the Theory Guide.

C2′-PS

(only for RSM) is the constant $C_2^{'}$ in Equation 4.211 in the Theory Guide.

C1-SSG-PS

(only for RSM with the **Quadratic Pressure-Strain Model**) is the constant C_1 in Equation 4.220 in the Theory Guide.

C1′-SSG-PS

(only for RSM with the **Quadratic Pressure-Strain Model**) is the constant C_1^* in Equation 4.220 in the Theory Guide.

C2-SSG-PS

(only for RSM with the **Quadratic Pressure-Strain Model**) is the constant C_2 in Equation 4.220 in the Theory Guide.

C3-SSG-PS

(only for RSM with the **Quadratic Pressure-Strain Model**) is the constant C_3 in Equation 4.220 in the Theory Guide.

C3′-SSG-PS

(only for RSM with the **Quadratic Pressure-Strain Model**) is the constant C_3^* in Equation 4.220 in the Theory Guide.

C4-SSG-PS

(only for RSM with the **Quadratic Pressure-Strain Model**) is the constant C_4 in Equation 4.220 in the Theory Guide.

C5-SSG-PS

(only for RSM with the **Quadratic Pressure-Strain Model**) is the constant C_5 in Equation 4.220 in the Theory Guide.

Prandtl Number

(only for the Spalart-Allmaras model) is the constant $\sigma_{\tilde{\nu}}$ in Equation 4.15 in the Theory Guide.

TKE Prandtl Number

(only for the standard or realizable k-ε model, the standard k-ω model, the k-kl-ω Transition model, or the RSM) is the effective "Prandtl" number for transport of turbulence kinetic energy σ_k. This effective Prandtl number defines the ratio of the momentum diffusivity to the diffusivity of turbulence kinetic energy via turbulent transport.

TKE (Inner) Prandtl #

(only for the BSL or SST k-ω model, and the Transition SST model) is the effective "Prandtl" number for the transport of turbulence kinetic energy, $\sigma_{k,1}$, inside the near-wall region. See Modeling the Turbulent Viscosity in the Theory Guide for details.

TKE (Outer) Prandtl #

(only for the BSL or SST k-ω model, and the Transition SST model) is the effective "Prandtl" number for the transport of turbulence kinetic energy, $\sigma_{k,2}$, outside the near-wall region. See Modeling the Turbulent Viscosity in the Theory Guide for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TDR Prandtl Number
is the effective "Prandtl" number for transport of the turbulent dissipation rate, σ_ε, for the standard or realizable k-ε model or the RSM. This effective Prandtl number defines the ratio of the momentum diffusivity to the diffusivity of turbulence dissipation via turbulent transport.

SDR (Inner) Prandtl #
(only for the BSL or SST k-ω model, and the Transition SST model) is the effective "Prandtl" number for the transport of the specific dissipation rate, $\sigma_{\omega,1}$, inside the near-wall region. See Modeling the Turbulent Viscosity in the Theory Guide for details.

SDR (Outer) Prandtl #
(only for the BSL or SST k-ω model, and the Transition SST model) is the effective "Prandtl" number for the transport of the specific dissipation rate, $\sigma_{\omega,2}$, outside the near-wall region. See Modeling the Turbulent Viscosity in the Theory Guide for details.

Dispersion Prandtl Number
(only for the k-ε multiphase models) is the effective "Prandtl" number for the dispersed phase, σ_{pq}. See Turbulence Models in the Theory Guide for details.

Energy Prandtl Number
(for any turbulence model except the RNG k-ε model) is the turbulent Prandtl number for energy, Pr_t, in Equation 4.228 in the Theory Guide. (This item will not appear for the LES model if the **Dynamic Energy Flux** option has been enabled, or for premixed or partially premixed combustion models.) For non-premixed and partially premixed combustion, the energy Prandtl Number is set equal to the PDF Schmidt Number.

Wall Prandtl Number
(for all turbulence models) is the turbulent Prandtl number at the wall, Pr_t in Equation 4.319 in the Theory Guide. (This item will not appear for adiabatic premixed combustion or partially premixed combustion models.)

Turbulent Schmidt Number
(for turbulent species transport calculations using any turbulence model except the RNG k-ε model) is the turbulent Schmidt number, Sc_t, in Equation 7.3 in the Theory Guide. (This item will not appear for the LES model if the **Dynamic Scalar Flux** option has been enabled.)

PDF Schmidt Number
(for non-premixed or partially premixed combustion calculations using any turbulence model) is the model constant σ_t in Equation 8.5 in the Theory Guide.

Production Limiter Clip Factor
This coefficient is used by the **Production Limiter**. For details, see Equation 4.382 in the *Fluent Theory Guide*.

User-Defined Transition Correlations
(only for the Transition SST model) allows you to select the user-defined correlations for **F_length**, **Re_thetac**, **Re_thetat**.

User-Defined Functions
allows you to select the user-defined functions for various constants.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Turbulent Viscosity
appears for Spalart Allmaras, k-ε, k-ω, DES, SDES with BSL / SST models, and SBES with BSL / SST. You can select the user-defined functions for turbulent viscosity in the drop-down list.

Prandtl Numbers, Prandtl and Schmidt Numbers
contains a list of relevant Prandtl and Schmidt numbers for which you can select user-defined functions.

TKE Prandtl Number
allows you to select a user-defined function to define the TKE Prandtl number for the standard and realizable k-ε models and the standard k-ω model.

TDR Prandtl Number
allows you to select a user-defined function to define the TDR Prandtl number for the standard and realizable k-ε models.

Energy Prandtl Number
allows you to select a user-defined function to define the turbulent Prandtl number for energy, when energy is enabled for all turbulence models except the RNG k-ε model. (This item will not appear for the LES model if the **Dynamic Energy Flux** option has been enabled.)

Wall Prandtl Number
allows you to select a user-defined function to define the turbulent Prandtl number at the wall for all turbulence models when energy is enabled.

Turbulent Schmidt Number
allows you to select a user-defined function to define the turbulent Schmidt number when species transport is enabled for all turbulence models except the RNG k-ε model. (This item will not appear for the LES model if the **Dynamic Scalar Flux** option has been enabled.)

SDR Prandtl Number
allows you to select a user-defined function to define the SDR Prandtl number for the standard k-ω model.

Subgrid-Scale Turbulent Viscosity
allows you to select a user-defined function for the subgrid-scale turbulent viscosity for the LES model.

Scale-Resolving Simulation Options
allows you to combine two turbulence models, which are applied on the appropriate regions of the flow domain. For additional information, see Scale-Resolving Simulation (SRS) Models (p. 756).

Scale-Adaptive Simulation (SAS)
allows you to apply SAS in combination with the following ω-based URANS models: the Standard or BSL k-ω model, the Transition SST model, and the ω-based Reynolds stress models. For additional information, see Setting Up Scale-Adaptive Simulation (SAS) Modeling (p. 781).

Detached Eddy Simulation (DES)
allows you to apply DES in combination with the BSL k-ω model and Transition SST model. For additional information, see Detached Eddy Simulation (DES) (p. 758).

Stress Blending (SBES) / Shielded DES
allows you to apply SBES or SDES in combination with the BSL k-ω model, SST k-ω model, and Transition SST model. See Stress-Blended Eddy Simulation (SBES) and Shielded Detached Eddy Simulation (SDES) in the Theory Guide for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Shielding Functions

allows you to select the shielding functions (**SST F1 Function**, **SST F2 Function**, **DDES**, and **IDDES**) for the Detached Eddy Simulation model with the BSL k-ω model, SST k-ω model, and Transition SST model. (see Shielding Functions for the BSL / SST / Transition SST Detached Eddy Simulation Model (p. 807) for details).

SBES / SDES Options

allows you to define the following settings as part of the SBES or SDES model:

Hybrid Model

specifies that the hybrid RANS-LES model used is either **SDES** (Shielded DES model), **SBES** (Stress-Blended Eddy Simulation model), or **SBES with User-Defined Function**. See Including the SDES or SBES Model with BSL, SST, and Transition SST Models (p. 805) for details.

Subgrid-Scale Model

contains options for the subgrid-scale model used in the LES region of your Stress-Blended Eddy Simulation. This portion of the dialog box will appear only if **SBES** or **SBES with User-Defined Function** is selected as the **Hybrid Model**.

Smagorinsky-Lilly

selects the Smagorinsky-Lilly subgrid-scale model described in Subgrid-Scale Models in the Theory Guide.

Dynamic Smagorinsky

selects the Smagorinsky-Lilly subgrid-scale model (described in Subgrid-Scale Models in the Theory Guide) with the dynamic stress model enabled.

WALE

selects the Wall-Adapting local Eddy-Viscosity model described in Wall-Adapting Local Eddy-Viscosity (WALE) Model in the Theory Guide.

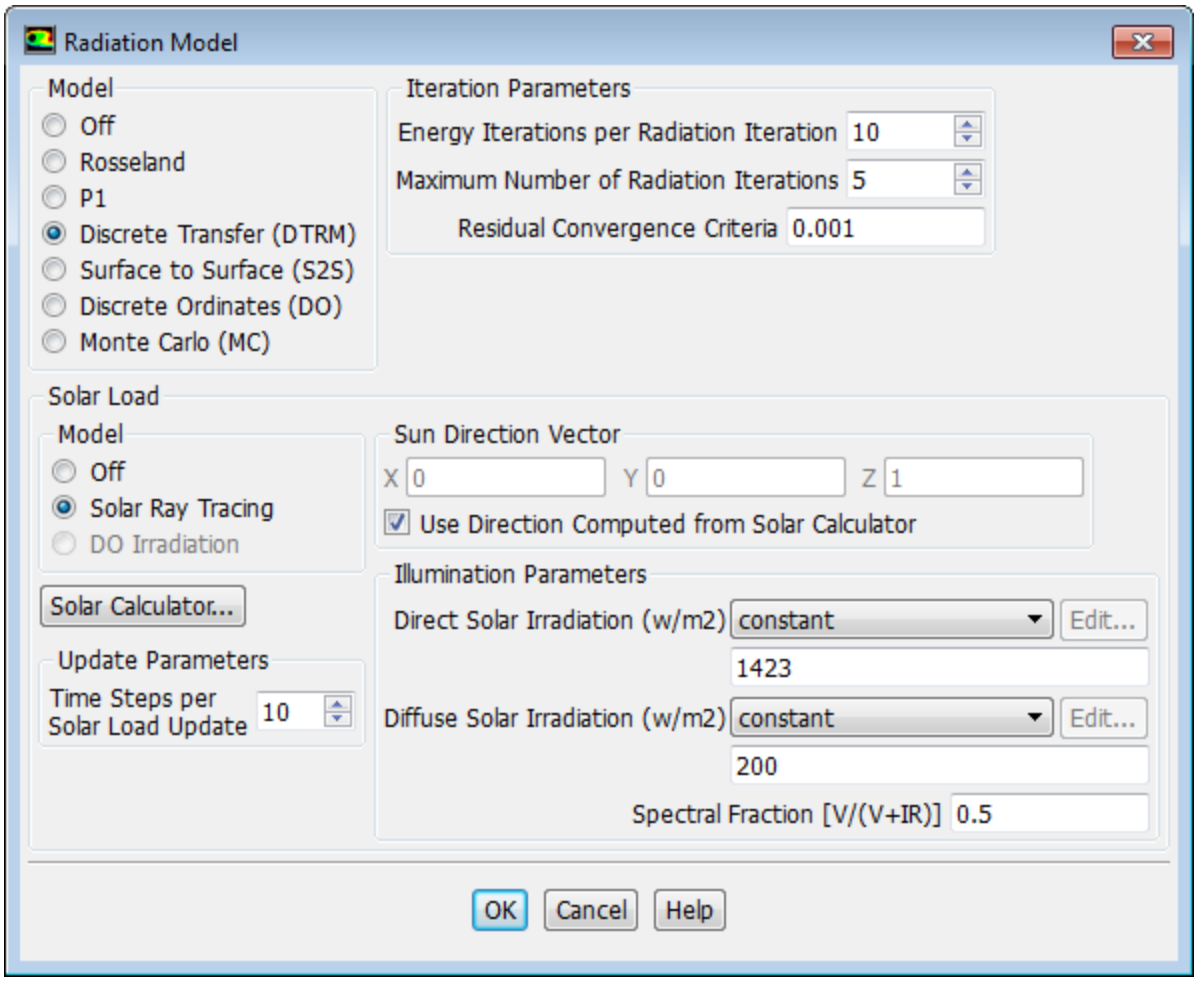
WMLES S-Omega

selects the algebraic Wall-Modeled LES S-Ω model described in Algebraic WMLES S-Omega Model Formulation in the Theory Guide.

37.4.4. Radiation Model Dialog Box

The **Radiation Model** dialog box allows you to select a model for radiation heat transfer and set the associated parameters. See Using the Radiation Models (p. 845) – Defining Non-Gray Radiation for the DO Model (p. 865) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Model

indicates which model, if any, is used to calculate radiation heat transfer. See Modeling Radiation (p. 845) for details about modeling radiation heat transfer.

Off

disables the calculation of radiation heat transfer.

Rosseland

enables the Rosseland radiation model.

P1

enables the P-1 radiation model.

Discrete Transfer (DTRM)

enables the discrete transfer radiation model (DTRM).

Surface to Surface (S2S)

enables the surface-to-surface (S2S) radiation model.

Discrete Ordinates (DO)

enables the discrete ordinates (DO) radiation model.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Monte Carlo (MC)
enables the Monte Carlo (MC) radiation model.

DO/Energy Coupling
allows you to couple the energy and DO intensity equations at each cell, solving them simultaneously. See Defining Non-Gray Radiation for the DO Model (p. 865) for details.

Iteration Parameters
contains parameters related to the DTRM, S2S, and the DO models. This portion of the dialog box will appear only if **Discrete Transfer (DTRM)**, **Surface to Surface (S2S)**, or **Discrete Ordinates (DO)** is selected as the **Model**.

Energy Iterations per Radiation Iteration
controls the frequency with which the radiation terms are updated as the continuous phase solution proceeds. The **Energy Iterations per Radiation Iteration** parameter is set to 10 by default. This implies that the radiation calculation is performed once every 10 iterations of the solution process. Increasing the number can speed the calculation process, but may slow overall convergence.

Maximum Number of Radiation Iterations
controls the maximum number of iterations of the radiation calculation during each global iteration. The default setting of 5 means that the radiosity will be updated up to 5 times. The actual number of iterations will be less if the residual convergence criterion is exceeded at any point during these iterations.

This item appears only when **Discrete Transfer** or **Surface to Surface** is selected as the **Model**.

Residual Convergence Criteria
determines when the radiation intensity update is converged. It is defined as the maximum normalized change in the surface intensity from one radiation iteration to the next (see Equation 12.9 (p. 881)).

This item appears only when **Discrete Transfer** or **Surface to Surface** is selected as the **Model**.

Target Number of Histories
determines the number of photon histories to be tracked for the Monte Carlo simulation. Too small of a number will produce "speckled" results. Increasing the target number of histories produces a smoother and more accurate solution, but at the expense of higher computation effort.

This item appears only when **Monte Carlo** is selected.

View Factors and Clustering
contains parameters related to the S2S model. This portion of the dialog box will appear only if **Surface to Surface** is selected as the **Model**. See Setting Up the S2S Model (p. 851) for more information about the use of these parameters.

Settings...
opens the View Factors and Clustering Dialog Box (p. 2194), in which you can set parameters related to surface clusters and view factors.

Compute/Write/Read...
allows you to compute the view factors, write the computed view factors to a file, and read the file into ANSYS Fluent. When you click **Compute/Write/Read...**, The Select File Dialog Box (p. 19) will open so that you can specify a name for the file where ANSYS Fluent should save the view factors after computing them, and indicate whether the files should be saved in binary format.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Read Existing File...
opens The Select File Dialog Box (p. 19), in which you can specify the file from which ANSYS Fluent should read view factors.

Angular Discretization
contains parameters for angular discretization and pixelation for the DO model. This portion of the dialog box will appear only if **Discrete Ordinates** is selected as the **Model**. See Setting Up the DO Model (p. 864) for more information about the use of these parameters.

Theta Divisions, Phi Divisions
define the number of control angles used to discretize each octant of the angular space (see Figure 5.3: Angular Coordinate System in the Theory Guide).

Theta Pixels, Phi Pixels
are used to control the pixelation that accounts for any control volume overhang (see Figure 5.7: Pixelation of Control Angle in the Theory Guide and the figures and discussion preceding it).

Radiation Mesh Options
This portion of the dialog box will appear only if the **Monte Carlo** model is selected.

Mesh Coarsening
Coarsens the radiation mesh used by the MC model. See Setting Up the MC Model (p. 867) for more information about the use of this parameter.

Non-Gray Model
contains parameters related to the non-gray P-1, DO, or MC model. This portion of the dialog box will appear only if **P1**, **Discrete Ordinates**, or **Monte Carlo** is selected as the **Model**. See Setting Up the P-1 Model with Non-Gray Radiation (p. 847) or Defining Non-Gray Radiation for the DO Model (p. 865) for more information about the use of these parameters.

Number of Bands
specifies the number of bands for the non-gray radiation.

Wavelength Intervals (n=1)
contains inputs that define each wavelength band. (It appears only when the **Number of Bands** is nonzero.) For each band, you can specify a **Name**, as well as the **Start** and **End** wavelength of the band in μm. Note that the wavelength bands are specified for vacuum (n=1). ANSYS Fluent will automatically account for the refractive index in setting band limits for media with n different from unity.

Solar Load
contains parameters related to solar load model that can be used to calculate radiation effects from the sun's rays that enter a computational domain. It is available only for the 3D version of ANSYS Fluent. See Solar Load Model (p. 886) for more information about the use of these parameters.

Model
indicates which model is used to calculate radiation effects.

Off
disables the calculation of solar radiations.

Solar Ray Tracing
enables the solar ray tracing algorithm

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DO Irradiation
enables the DO irradiation option. Select **Discrete Ordinates** under **Model** before selecting this option.

Solar Calculator
opens the Solar Calculator Dialog Box (p. 2199).

Sun Direction Vector
contains the components of sun direction vector.

X, Y, Z
are the components of the sun direction vector.

Use Direction Computed from Solar Calculator
enables the use of direction vector computed from the solar calculator.

Illumination Parameters
contains illumination options.

Direct Solar Irradiation
is the amount of energy per unit area due to direct solar irradiation.

Diffuse Solar Irradiation
is the amount of energy per unit area due to diffuse solar irradiation.

Spectral Fraction
is the fraction of incident solar radiation in the visible part of the solar radiation spectrum. The spectral fraction is not used for DO irradiation since the DO implementation is intended only for a single band. This parameter is available only for **Solar Ray Tracing**.

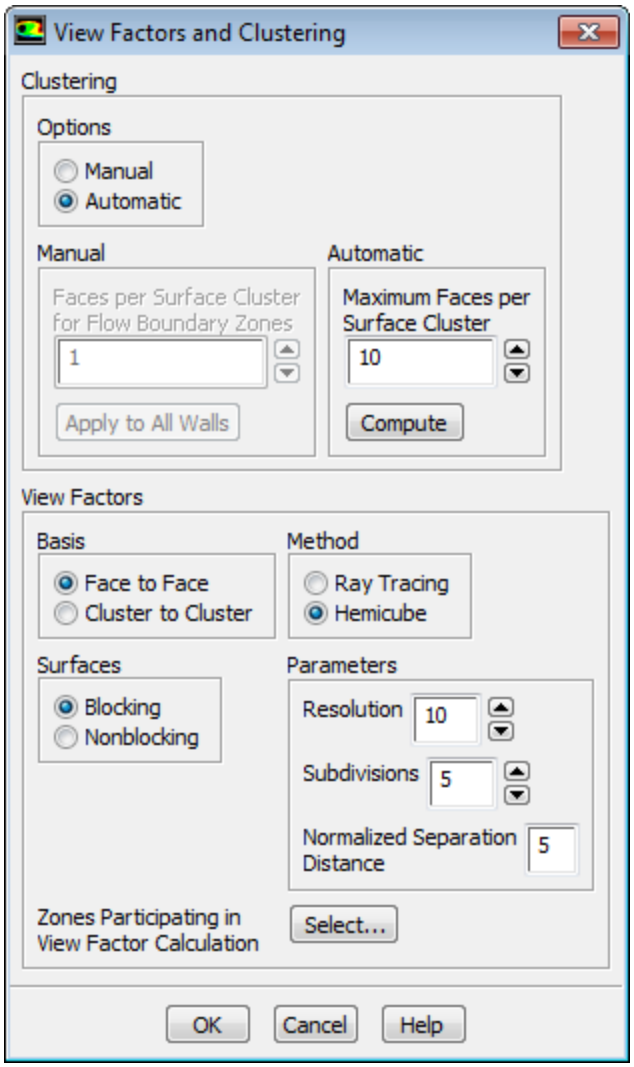
Update Parameters
contains update parameters for transient simulations.

Time Steps per Solar Load Update
specifies the number of time steps that will direct the ANSYS Fluent solver to update the solar load data for the specified flow-time intervals in the unsteady solution process.

37.4.5. View Factors and Clustering Dialog Box

The **View Factors and Clustering** dialog box allows you to set parameters related to surface clusters and view factors for the surface-to-surface radiation model. See Setting Up the S2S Model (p. 851) for information about using this dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Clustering

contains the methods and settings for creating surface clusters. See Forming Surface Clusters (p. 853) for information about surface cluster settings.

Options

gives you a choice of forming clusters either manually or automatically.

Manual

allows you to form clusters manually.

Automatic

allows ANSYS Fluent to form the cluster automatically.

Manual

(available only when the **Manual** option is enabled) allows you to set the faces per surface cluster (FPSC) value for walls and inlet and outlet boundaries.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Faces per Surface Cluster for Flow Boundary Zones

specifies the number of faces in each surface cluster for inlet and outlet boundaries (that is, exhaust fan, inlet vent, intake fan, outlet vent, mass-flow inlet, mass-flow outlet, pressure far-field, pressure inlet, pressure outlet, outflow, and velocity inlet boundaries). This FPSC value can also be applied to all walls that are adjacent to fluid zones by clicking the **Apply to All Walls** button. The default is set to `1`. See Clustering in the Theory Guide for details about clustering.

Apply to All Walls

applies the value specified in the **Faces per Surface Cluster for Flow Boundary Zones** field to all walls that are adjacent to fluid zones.

Automatic

(available only when the **Automatic** option is enabled) allows you to assign different faces per surface cluster (FPSC) values to the walls automatically, based on the distance of the walls from and the FPSC values of the walls that are defined as critical.

Maximum Faces per Surface Cluster

specifies the maximum number of faces per surface cluster automatically assigned to non-critical wall zones adjacent to fluid zones. The default is set to `10`.

Compute

results in ANSYS Fluent automatically calculating and updating the face per surface cluster values in the boundary conditions dialog box for non-critical wall zones adjacent to fluid zones, without computing the clusters.

View Factors

contains settings for computing the view factors. See Setting Up the View Factor Calculation (p. 856) for information about view factor calculation settings.

Basis

specifies how surfaces are defined for the calculation of view factors. See Selecting the Basis for Computing View Factors (p. 856) for more information.

Face to Face

specifies that the surfaces used to calculate the view factors are the boundary faces.

Cluster to Cluster

specifies that the surfaces used to calculate the view factors are the clusters defined by the settings in the **Clustering** group box. The cluster to cluster basis is only available for 3D cases.

Surfaces

specifies the geometric orientation of surface pairs with respect to each other when using the hemicube method.

Blocking

specifies that the view factor calculation accounts for surfaces that block the views between the surfaces under consideration.

Nonblocking

specifies that the view factor calculation does not account for surfaces that block the views between the surfaces under consideration.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Method
specifies the method for computing the view factors. See Selecting the Method for Computing View Factors (p. 857) for information about choosing a method.

Hemicube
specifies the use of the hemicube method for computing the view factors. The hemicube method is available only for 3D and axisymmetric cases, and should not be used if any of the zones are defined as periodic or symmetry boundaries.

Ray Tracing
specifies the use of the ray tracing method for computing the view factors. For 2D cases, the number of rays used with this method is two times the value set for **Resolution** in the **Parameters** group box; for 3D cases, the number of rays is three times the square of the **Resolution** value.

Parameters
contains settings related to the hemicube and ray tracing methods for computing the view factors. All of these inputs are available when **Hemicube** is selected under **Method**, whereas only **Resolution** is available when **Ray Tracing** is selected. See Selecting the Method for Computing View Factors (p. 857) for information about setting the method parameters.

Resolution
specifies the resolution of the hemicube. The default value is set to `10`. You can increase the value to reduce aliasing effects that can lead to overestimated or underestimated view factors.

Subdivisions
specifies the number of subfaces into which each face is divided. The default value is set to 5. This parameter is only available when the hemicube method is used in conjunction with the face to face basis.

Normalized Separation Distance
specifies the ratio of the minimum face separation to the effective diameter of the face. The default value is set to 5. This parameter is only available for the hemicube method.

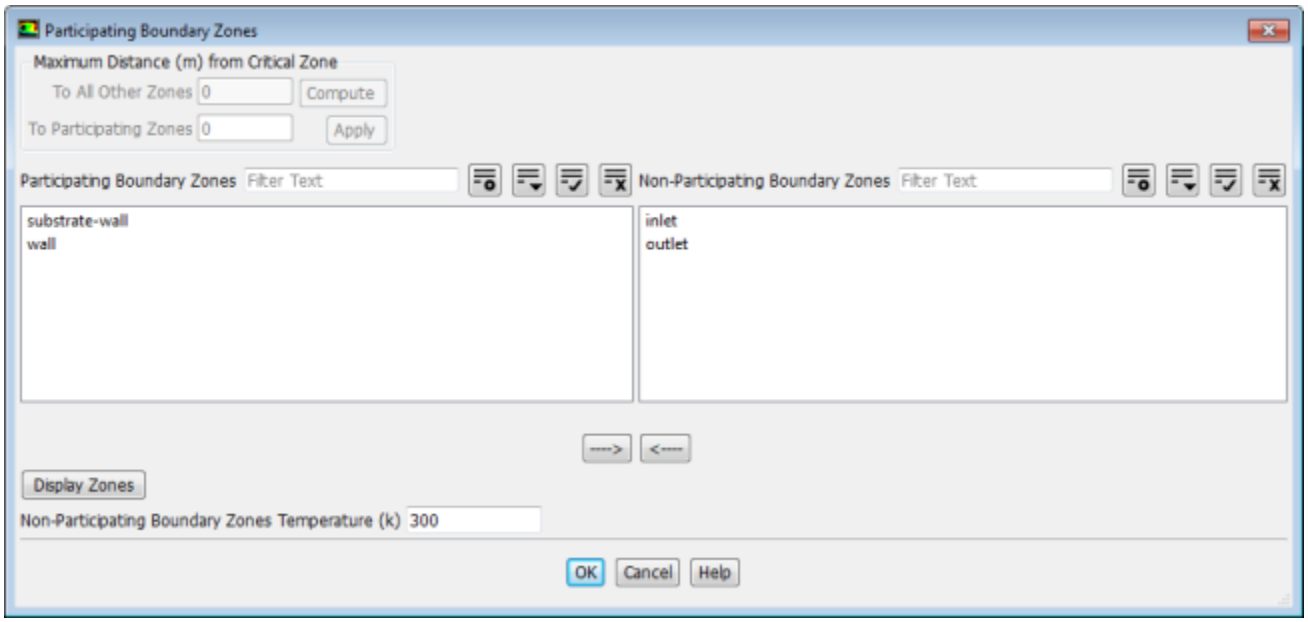
Zones Participating in View Factor Calculation
allows you to define which boundary zones participate in the view factor calculation. See Specifying Boundary Zone Participation (p. 858) for more information.

Select...
opens the Participating Boundary Zones Dialog Box (p. 2197), where you can define which boundary zones participate in the view factor calculation.

37.4.6. Participating Boundary Zones Dialog Box

The **Participating Boundary Zones** dialog box allows you to define which boundary zones participate in the view factor calculation, to display zones in the graphics window, and set the temperature of zones that do not participate in the view factor calculation. See Specifying Boundary Zone Participation (p. 858) for details. This dialog box opens when you click the **Select...** button next to the **Zones Participating in View Factor Calculation** label in the **View Factors and Clustering** dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Maximum Distance from Critical Zone
allows you to view the maximum distance between critical zones and other zones, and set all boundary zones that are located beyond a certain distance from a critical zone as not participating in the view factor calculation.

To All Other Zones
displays the maximum of the distances between the centroids of critical and all other wall, inlet, and exit zones when the **Compute** button is clicked. This field is not editable. This is only available when using the **Automatic** option for clustering.

Compute
calculates the distances between the centroids of critical zones and all the other wall, inlet, and exit zones and displays the maximum value in the **To All Other Zones** field and the **To Participating Zones** text-entry box. It requires the definition of the critical zone.

To Participating Zones
displays the maximum of the distances between the centroids of critical and all other wall, inlet, and exit zones when the **Compute** button is clicked. Unlike the **To All Other Zones** field, you can edit this text-entry box. You can specify the maximum distance allowed between the centroids of critical zones and all other wall, inlet, or exit zones that participate in the view factor calculation; when you click the **Apply** button, all of the zones will be marked as either participating or not participating, according to the distance criteria you specified. Note that this field is only available when using the **Automatic** option for clustering.

Apply
marks all of the wall, inlet, and exit zones as either participating or not participating in the view factor calculation, depending on whether the distance from their centroid to the centroid of a critical zone is equal to or less than value entered for **To Participating Zones**. The **Participating Boundary Zones** and **Non-Participating Boundary Zones** lists will be updated accordingly.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Participating Boundary Zones
shows all the zones that are participating in the view factor calculation. You can select a zone and click the arrow button that points to the right to move the zone to the **Non-Participating Boundary Zones** list.

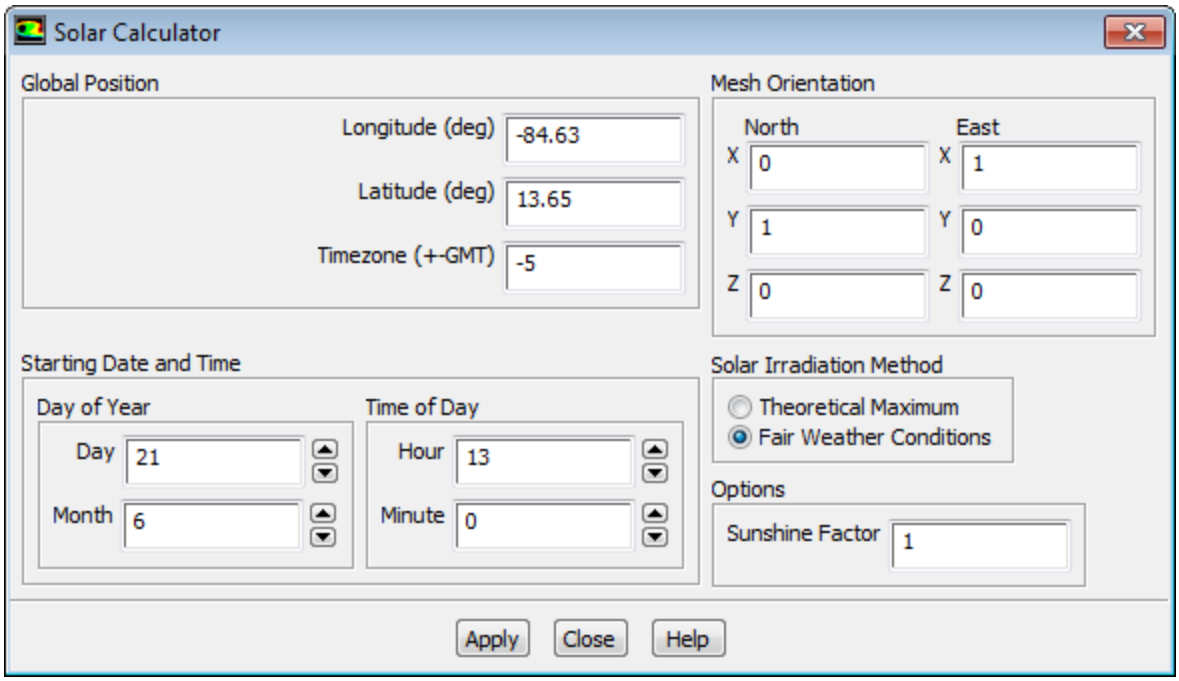
Non-Participating Boundary Zones
shows all the zones that are not participating in the view factor calculation. You can select a zone and click the arrow button that points to the left to move the zone to the **Participating Boundary Zones** list.

Display Zones
displays any zones selected in the **Participating Boundary Zones** and **Non-Participating Boundary Zones** lists in the graphics window.

Non-Participating Boundary Zones Temperature
allows you to specify the temperature of the zones that do not participate in the view factor calculation.

37.4.7. Solar Calculator Dialog Box

The **Solar Calculator** dialog box allows you to set parameters related to the calculation of solar load models. See Solar Load Model (p. 886) for details. This dialog box opens when you click the **Solar Calculator...** button in the **Radiation Model** dialog box.



Controls

Global Position
contains the parameters to define the position of solar radiation.

Longitude
specifies the longitude of the desired location in degrees. Values may range from -180 to 180, where negative values indicate the Western hemisphere and positive values indicate the Eastern hemisphere.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Latitude
specifies the latitude of the desired location in degrees. Values may range from -90 (South pole) to 90 (North pole) with 0 defined as the equator.

Timezone
specifies the local time zone of the desired location in hours relative to Greenwich Mean Time (+-GMT). This integer value can range from 12 to -12.

Starting Date and Time
contains parameters to specify date and time.

Day of Year
contains parameters to specify day and month.

Time of Day
contains parameters to specify hour and minutes.

Mesh Orientation
specifies the orientation as the vectors for North and East in the CFD grid system of coordinates.

Solar Irradiation Method
contains parameters to choose the solar irradiation method.

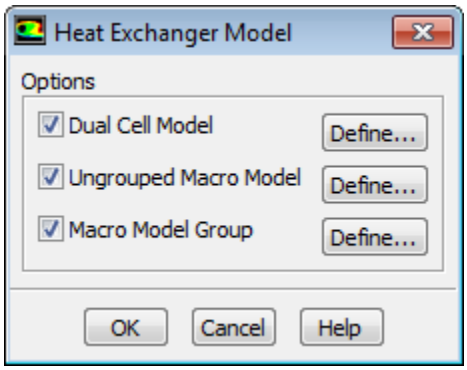
Theoretical Maximum
enables theoretically maximum solar irradiation method.

Fair Weather Conditions
enables fair weather condition solar irradiation method.

Sunshine Factor
is a linear reduction factor for the computed incident load that allows for cloud cover to be accounted for, if appropriate.

37.4.8. Heat Exchanger Model Dialog Box

The **Heat Exchanger Model** dialog box allows you to define a heat exchanger as part of your model. See Modeling Heat Exchangers (p. 919) for details about using the items below.



Controls

Options
contains options related to the heat exchanger model. See Choosing a Heat Exchanger Model (p. 920) for details about the differences between the models associated with these options.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

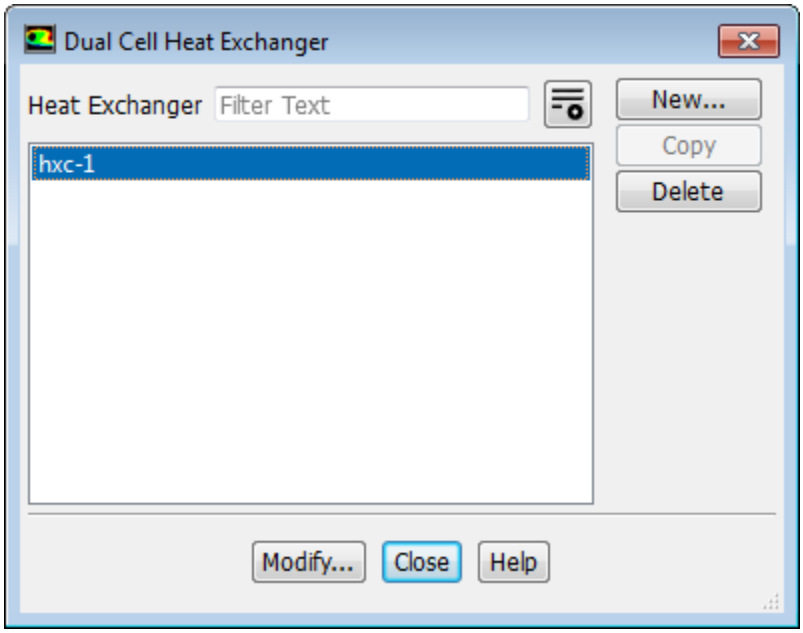
Dual Cell Model
allows you to enable or disable the Dual Cell heat exchanger model. When this option is enabled, the corresponding **Define...** button opens the Dual Cell Heat Exchanger Dialog Box (p. 2201).

Ungrouped Macro Model
allows you to enable or disable the Ungrouped Macro heat exchanger model. When this option is enabled, the corresponding **Define...** button opens the Ungrouped Macro Heat Exchanger Dialog Box (p. 2207).

Macro Model Group
allows you to enable or disable the Macro heat exchanger group model. When this option is enabled, the corresponding **Define...** button opens the Macro Heat Exchanger Group Dialog Box (p. 2212).

37.4.9. Dual Cell Heat Exchanger Dialog Box

The **Dual Cell Heat Exchanger** dialog box allows you to use the NTU method for heat transfer calculations. This model allows the solution of auxiliary flow on a separate mesh (other than the primary fluid mesh). See Using the Dual Cell Heat Exchanger Model (p. 922) for information about using this dialog box.



Controls

Heat Exchanger
contains a list of predefined heat exchangers.

New...
opens the Set Dual Cell Heat Exchanger Dialog Box (p. 2202).

Copy
opens the Copy From Dialog Box (p. 2206), which allows you to copy the currently selected heat exchanger.

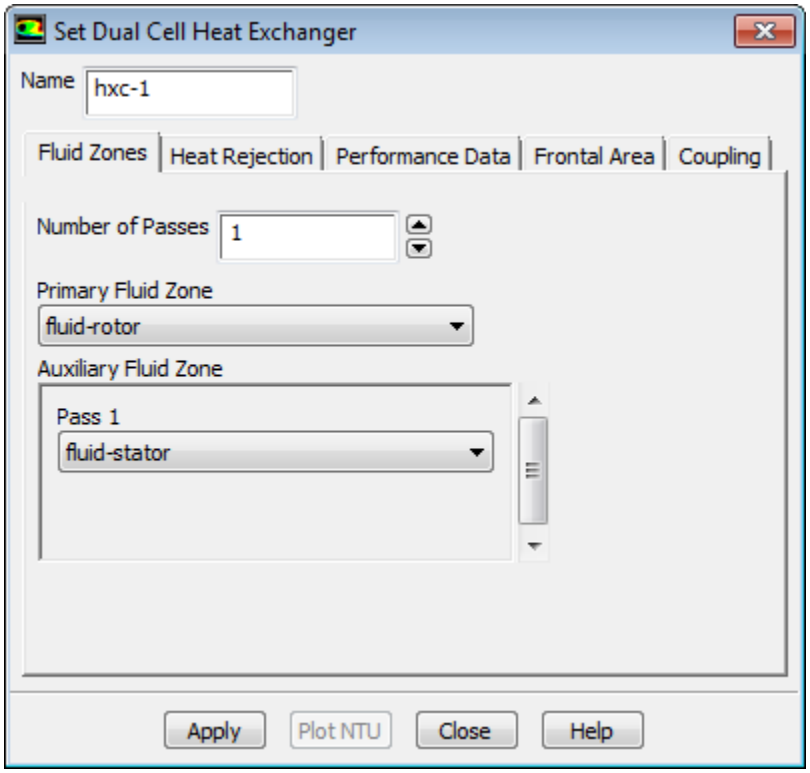
Delete
removes the currently selected heat exchanger.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modify...
opens the Set Dual Cell Heat Exchanger Dialog Box (p. 2202).

37.4.10. Set Dual Cell Heat Exchanger Dialog Box

The **Set Dual Cell Heat Exchanger** dialog box allows you to define the heat exchanger parameters. See Using the Dual Cell Heat Exchanger Model (p. 922) for details about using this dialog box.



Controls

Name
allows you to specify a name for the dual cell heat exchanger.

Fluid Zones
allows you to specify the fluid zone parameters for the heat exchanger.

Number of Passes
specifies the number of passes for the heat exchanger.

Primary Fluid Zone
allows you to specify the fluid of the primary fluid zone for the heat exchanger.

Auxiliary Fluid Zone
allows you to specify the fluid for the auxiliary fluid zone, per pass.

Important

The selected zones must be overlapping in physical space.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Heat Rejection
contains parameters specific to the rejection of heat in the heat exchanger.

Options
allows you to specify how the heat rejection in the heat exchanger is computed.

Fixed Heat Rejection
allows you to specify heat rejection parameters.

Fixed Inlet Temperature
allows you to use total heat rejection as the desired output.

Heat Rejection Targeted
allows you to specify the heat rejection desired from the heat exchanger (available only when the **Fixed Heat Rejection** option is enabled).

Inlet Zone for Temperature Updates
allows ANSYS Fluent to change the temperature of the specified inlet zone in order to match the targeted heat rejection (available only when the **Fixed Heat Rejection** option is enabled).

Temperature Update Under-Relaxation
controls convergence (available only when the **Fixed Heat Rejection** option is enabled).

Iteration Interval Between Temperature Updates
controls divergence (available only when the **Fixed Heat Rejection** option is enabled).

Performance Data
contains parameters for specifying the heat exchanger's performance data.

Options
allows you to choose between using raw performance data or NTU performance data.

Raw Data
allows you to specify raw performance data for the heat exchanger.

NTU Data
allows you to specify NTU performance data for the heat exchanger.

Heat Exchanger Performance Data
contains parameters concerning the heat exchanger's performance data.

NTU Table
opens the NTU Table Dialog Box (p. 2205) (available only when the **NTU Data** option is enabled).

Heat Transfer Table
opens the Heat Transfer Data Table Dialog Box (p. 2204) (available only when the **Raw Data** option is enabled).

Effectiveness-NTU Relation
computes the NTU values from the heat transfer data. Choose **cross-flow-unmixed**, **parallel-flow**, or **counter-flow**, all of which are described in NTU Relations (available only when the **Raw Data** option is enabled).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reference Inlet Temperature
allows you to specify the inlet reference temperature for the primary and the auxiliary fluids. (Available only when the **Raw Data** option is enabled.)

Frontal Area
allows you to specify the frontal area for the heat exchanger.

Primary Fluid
allows you to specify a value for the **Core Frontal Area** for the primary fluid, or to compute the value from a surface zone using the **Compute From** drop-down list.

Auxiliary Fluid
allows you to specify a value for the **Core Frontal Area** for the auxiliary fluid, or to compute the value from a surface zone using the **Compute From** drop-down list.

Coupling
specifies parameters when you want to couple the heat exchanger passes.

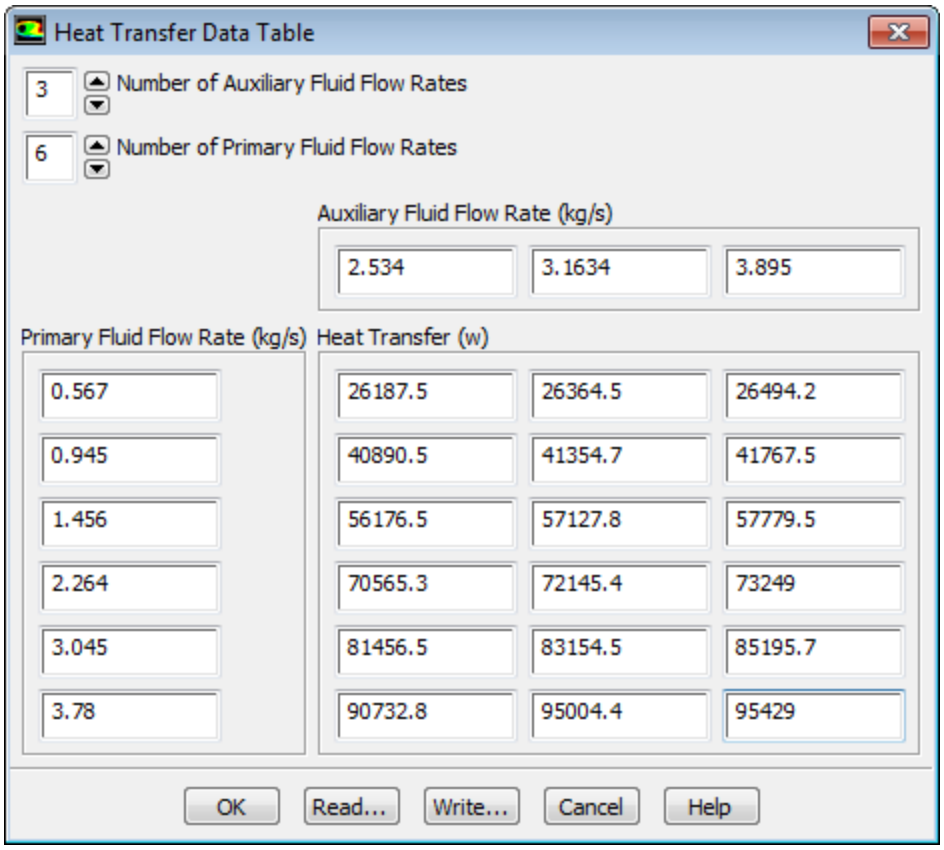
Temperature
specifies, by default, the mass-weighted average for the temperature of the outlet of Pass 1 to the inlet of Pass 2. Similarly, the mass-weighted-average temperature of the outlet of Pass 2 will be applied at the inlet zone of Pass 3, and so on. (Available only when multiple passes are specified in the **Fluid Zones** tab.)

Plot NTU
plots the performance data curve for the selected heat exchanger. The performance data is supplied through the **Performance Data** tab.

37.4.11. Heat Transfer Data Table Dialog Box

The **Heat Transfer Data Table** dialog box contains information on the number of fluid flow rates and heat transfer data for the primary and auxiliary fluids. See Using the Ungrouped Macro Heat Exchanger Model (p. 932) for details about using this dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Number of Auxiliary Fluid Flow Rates
sets the number of auxiliary fluid flow rates.

Number of Primary Fluid Flow Rates
sets the number of primary fluid flow rates.

Auxiliary Fluid Flow Rate
sets fluid flow rates for the auxiliary fluid.

Primary Fluid Flow Rate
sets fluid flow rates for the primary fluid.

Heat Transfer
sets the heat transfer for the corresponding primary and auxiliary fluid flow rates.

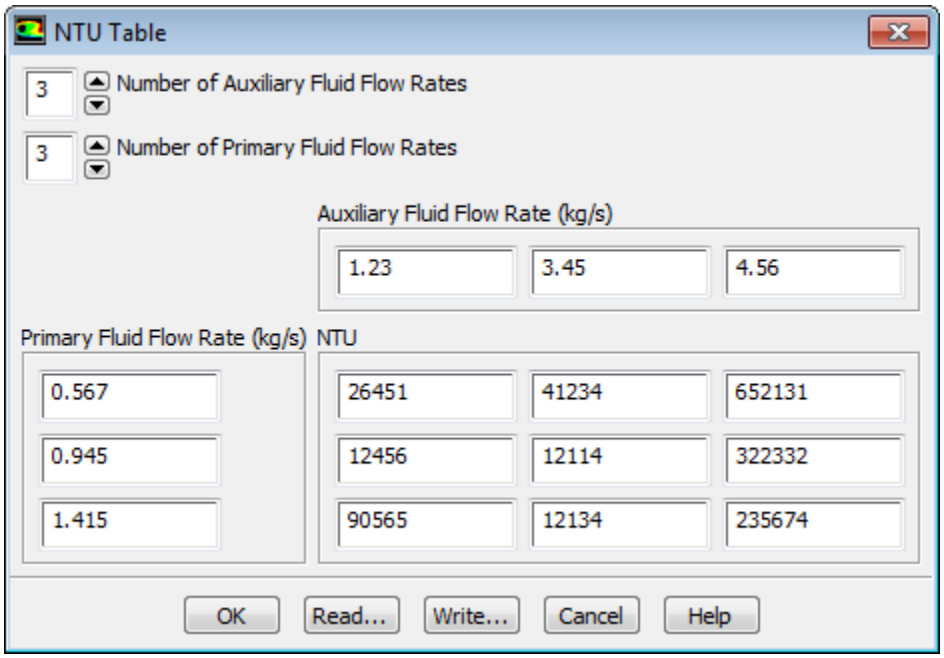
Read...
allows you to read in a file containing heat transfer data.

Write...
allows you to write a file containing heat transfer data.

37.4.12. NTU Table Dialog Box

The **NTU Table** dialog box contains information on the number of fluid flow rates and NTU data for the primary and auxiliary fluids. See Using the Ungrouped Macro Heat Exchanger Model (p. 932) for details about using this dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Number of Auxiliary Fluid Flow Rates
sets the number of auxiliary fluid flow rates.

Number of Primary Fluid Flow Rates
sets the number of primary fluid flow rates.

Auxiliary Fluid Flow Rate
sets fluid flow rates for the auxiliary fluid.

Primary Fluid Flow Rate
sets fluid flow rates for the primary fluid.

NTU
sets the NTU values for the corresponding primary and auxiliary fluid flow rates.

Read...
allows you to read in a file containing NTU data.

Write...
allows you to write a file containing NTU data.

37.4.13. Copy From Dialog Box

The **Copy From** dialog box allows you to copy the setup of one heat exchanger to another.

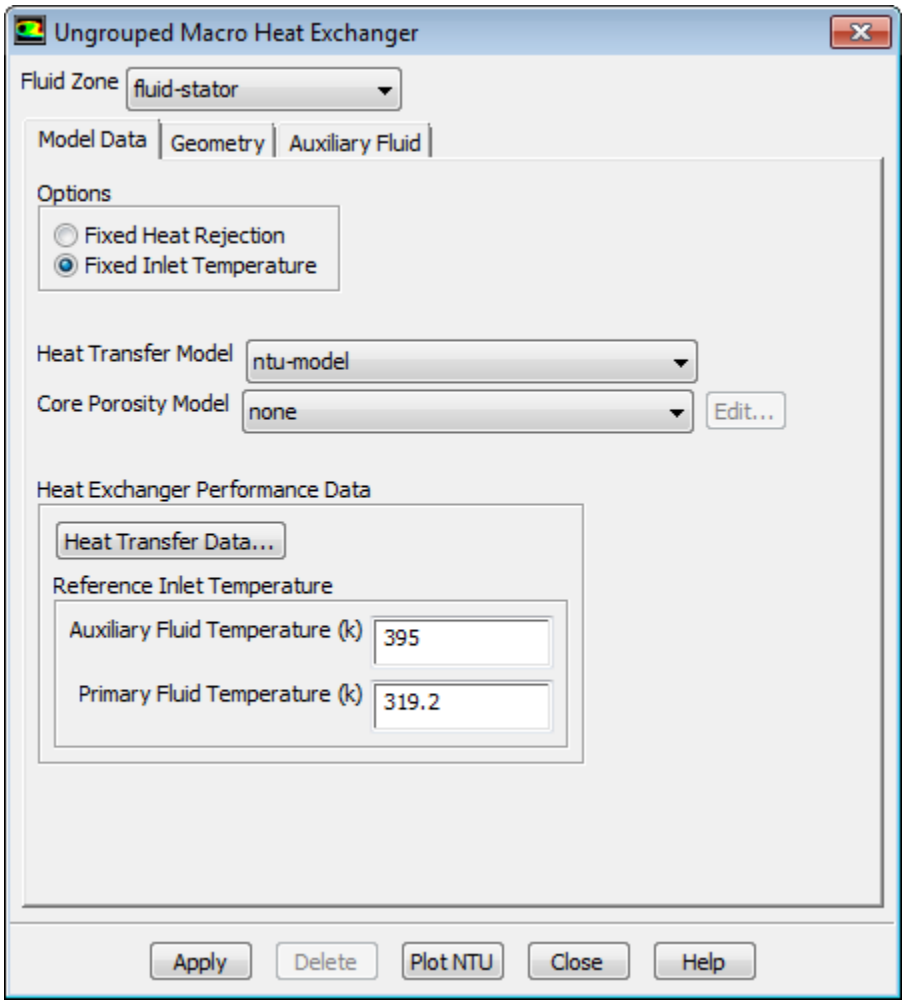
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Existing Heat Exchangers
contains a list of heat exchangers, from which you can copy the settings from one heat exchanger to another.

37.4.14. Ungrouped Macro Heat Exchanger Dialog Box

The **Ungrouped Macro Heat Exchanger** dialog box allows you to set up the ungrouped macro heat exchanger model. See Using the Ungrouped Macro Heat Exchanger Model (p. 932) for details about using this dialog box.



Controls

Fluid Zone
specifies the zone that represents the heat exchanger.

Model Data
contains the parameters related to the heat exchanger model.

Options
allows you to choose one of the following settings:

Fixed Heat Rejection
specifies that ANSYS Fluent should compute the auxiliary fluid inlet temperature for a specified heat rejection.

Fixed Inlet Temperature
specifies that ANSYS Fluent should compute the total heat rejection of the core for a given inlet auxiliary temperature.

Heat Transfer Model
allows you to specify either the **ntu-model** or the **simple-effectiveness-model** for heat transfer. See Choosing a Heat Exchanger Model (p. 920) for information on the differences between these models.

Core Porosity Model
contains a drop-down list of all available core porosity models.

Edit...
opens the Core Porosity Model Dialog Box (p. 2211).

Heat Exchanger Performance Data
contains the parameters for heat transfer.

Heat Transfer Data...
opens the Heat Transfer Data Table Dialog Box (p. 2204). This item will appear for the NTU model only.

Auxiliary Fluid Temperature
specifies the auxiliary fluid temperature. This item will appear only for the NTU model.

Primary Fluid Temperature
specifies the gas stream temperature. This item will appear only for the NTU model.

Velocity Effectiveness Curve...
opens the Velocity Effectiveness Curve Dialog Box (p. 2210) in which you can define the effectiveness of the heat exchanger core (ε in Equation 6.11 in the Theory Guide). This item will appear for simple effectiveness model only.

Geometry
contains parameters to define the macro grid.

Width
sets the width of the heat exchanger core. The **Width** is measured in the pass-to-pass direction.

Height
sets the height of the heat exchanger core. The **Height** is measured in the auxiliary fluid inlet direction.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Depth
sets the depth of the heat exchanger core.

Number of Passes
specifies the number of passes for the macro grid. (See Figure 13.17: 1x4x3 Macros (p. 939).)

Number of Rows/Pass, Number of Columns/Pass
specify the number of macro rows and columns per pass in the macro grid. (See Figure 13.17: 1x4x3 Macros (p. 939).)

View Passes
displays the macro grid. (This button becomes available after you click **Apply**.) The path of the auxiliary fluid is color-coded in the display: macro 0 is red and macro $n-1$ is blue.

Draw Mesh
toggles between displaying and not displaying the mesh when the macro mesh is displayed (using the **View Passes** button). The Mesh Display Dialog Box (p. 2163) opens when **Draw Mesh** is selected.

Update from Plane Tool
updates the **Auxiliary Fluid Inlet Direction** and **Pass-to-Pass Direction** from the plane tool orientation. The **Width**, **Height**, and **Depth** will also be updated. See Using the Plane Tool (p. 1724) for information about using the plane tool.

Auxiliary Fluid Inlet Direction (height)
specifies the direction in which the auxiliary fluid enters the heat exchanger. See Figure 13.17: 1x4x3 Macros (p. 939).

Pass-to-Pass Direction (width)
specifies the direction in which the auxiliary fluid moves at the end of each pass through the heat exchanger. See Figure 13.17: 1x4x3 Macros (p. 939).

Auxiliary Fluid
contains the option to specify the auxiliary fluid properties.

Auxiliary Fluid Properties Method
contains options for specifying auxiliary fluid properties.

constant-specific-heat
allows you to specific a constant value for the auxiliary fluid specific heat.

user-defined-enthalpy
allows you to specify a user-defined function for the auxiliary fluid enthalpy.

Auxiliary Fluid Specific Heat
specifies the value of $c_{p,auxiliary}$ in Equation 6.17 in the Theory Guide. This value is specified only if **constant-specific-heat** is selected.

Auxiliary Fluid Enthalpy UDF
allows you to specify a user-defined function for the auxiliary fluid enthalpy (see Equation 6.17 in the Theory Guide). This option is available only when **user-defined-enthalpy** is selected.

Auxiliary Fluid Flow Rate
sets the flow rate of the auxiliary fluid ($\dot{m}$ in Equation 6.16 in the Theory Guide).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Heat Rejection
sets the total heat rejection (q_{total} in Equation 6.15 in the Theory Guide). This value is specified only if **Fixed Heat Rejection** is selected.

Initial Temperature
sets an initial guess for the inlet temperature (T_{in} in Equation 6.11 and Equation 6.16 in the Theory Guide). This value is specified only if **Fixed Heat Rejection** is selected.

Inlet Temperature
sets the auxiliary fluid initial temperature (T_{in} in Equation 6.11 and Equation 6.16 in the Theory Guide). This value is specified only if **Fixed Inlet Temperature** is selected in the **Model Data** tab.

Inlet Pressure
sets the auxiliary fluid inlet pressure. This value is specified only if **user-defined-enthalpy** is selected.

Inlet Quality
specifies the value of x in Equation 6.20 in the Theory Guide. This value is specified only if **user-defined-enthalpy** is selected.

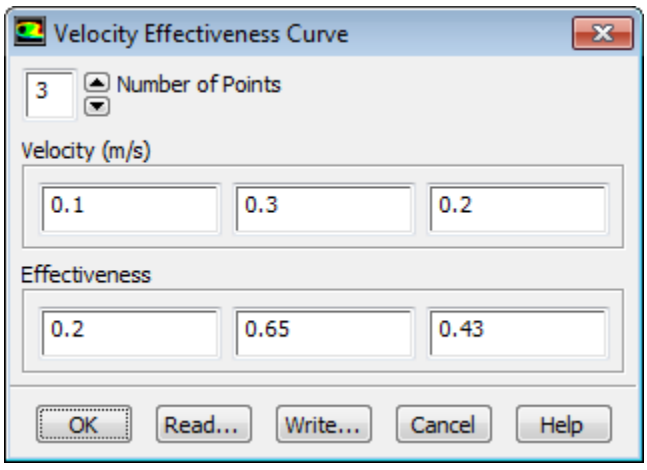
Pressure Drop
specifies the value of p_{in} in Equation 6.21 in the Theory Guide. This value is specified only if **user-defined-enthalpy** is selected.

Apply
saves all the settings for the heat exchanger specified in the **Fluid Zone** list.

Delete
deletes the heat exchanger specified in the **Fluid Zone** list.

37.4.15. Velocity Effectiveness Curve Dialog Box

The **Velocity Effectiveness Curve** dialog box allows you to define effectiveness curve. It is opened by clicking **Velocity Effectiveness Curve...** in the Ungrouped Macro Heat Exchanger Dialog Box (p. 2207).



Controls

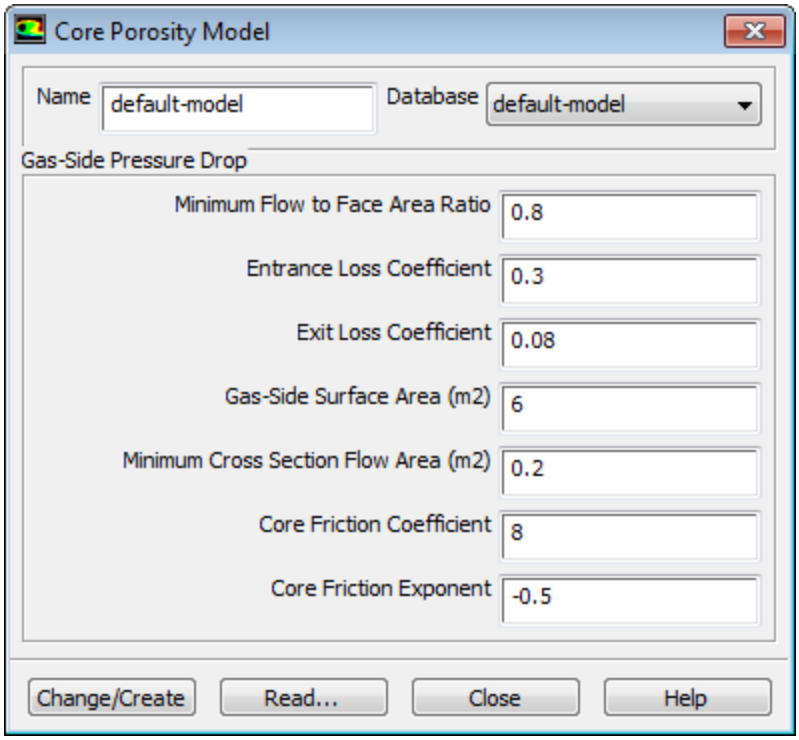
Number of Points
specifies the number of data pairs in the effectiveness profile. The default value of 1 indicates a constant effectiveness.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Velocity, Effectiveness
specify the data pairs for the effectiveness profile. These items are available only if the simple effective model has been selected. Note that the **Effectiveness** values must be within the range of 0–1.

37.4.16. Core Porosity Model Dialog Box

The **Core Porosity Model** dialog box allows you to modify or define a heat exchanger core model. This dialog box opens when you click **Edit...** in the Ungrouped Macro Heat Exchanger Dialog Box (p. 2207).



Controls

Name
specifies the name of a new heat exchanger core model.

Database
contains a drop-down list of all heat exchanger core models that are currently available.

Gas-Side Pressure Drop
contains parameters that define the air-side pressure drop.

Minimum Flow to Face Area Ratio
sets the value of σ in Equation 6.2 in the Theory Guide.

Entrance Loss Coefficient
sets the value of K_c in Equation 6.2 in the Theory Guide.

Exit Loss Coefficient
sets the value of K_e in Equation 6.2 in the Theory Guide.

Gas Side Surface Area
sets the value of A in Equation 6.2 in the Theory Guide.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Minimum Cross Section Flow Area
sets the value of A_c in Equation 6.2 in the Theory Guide.

Core Friction Coefficient
sets the value of a in Equation 6.3 in the Theory Guide.

Core Friction Exponent
sets the value of b in Equation 6.3 in the Theory Guide.

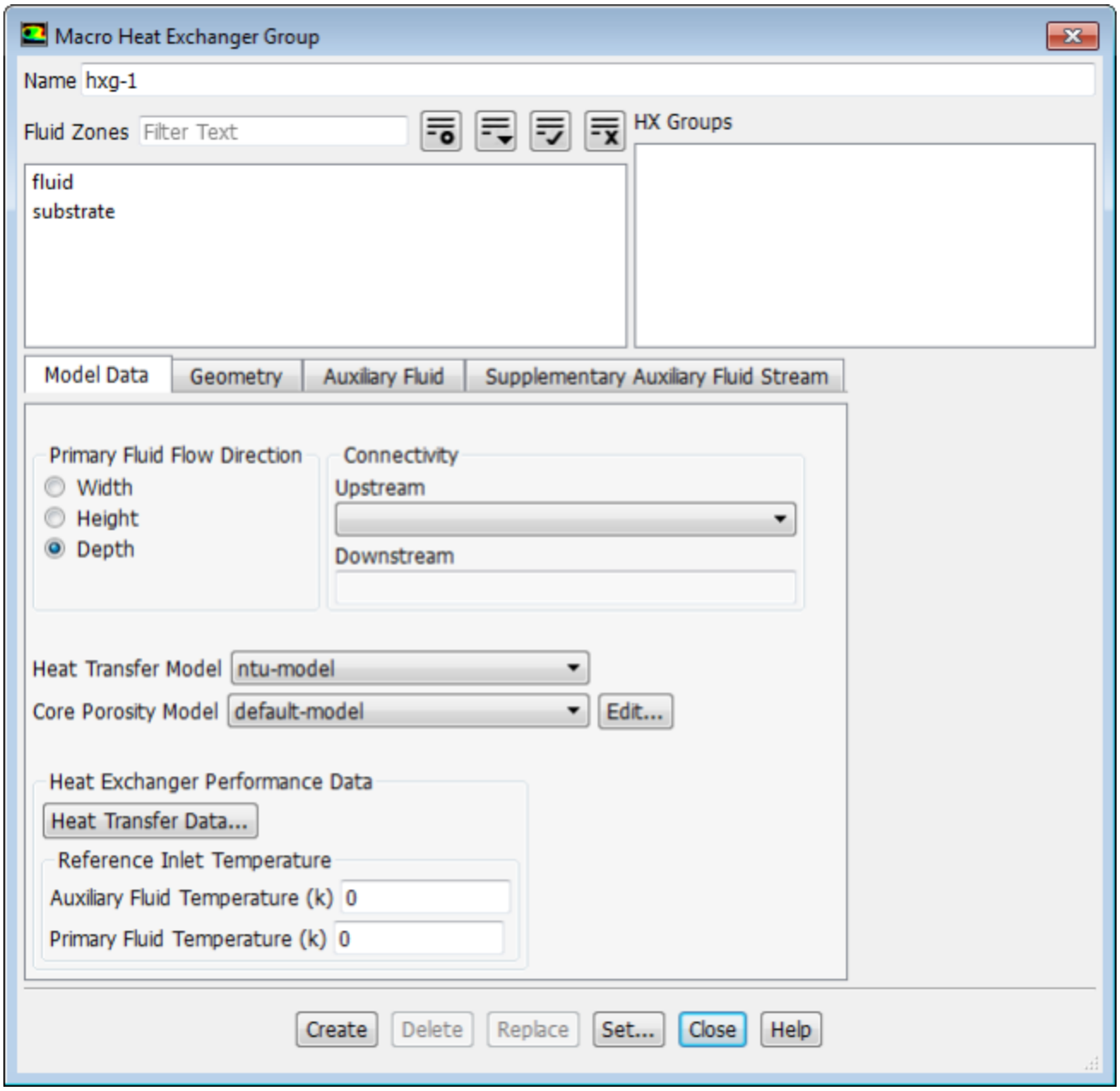
Change/Create
saves the settings in the dialog box and adds the new model to the **Database** list.

Read...
opens The Select File Dialog Box (p. 19), in which you can select an external file containing a predefined heat exchanger core model.

37.4.17. Macro Heat Exchanger Group Dialog Box

The **Macro Heat Exchanger Group** dialog box allows you to modify or define a heat exchanger core group.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name
specifies the name of a new heat exchanger group.

Fluid Zones
contains a list of all the fluid zones.

HX Groups
contains list of the heat exchanger groups.

Model Data
contains all the parameters to be specified for the model.

Primary Fluid Flow Direction
gives you a choice of gas flow direction.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Width, Height, Depth
specifies the width, height and the depth of the gas flow direction.

Connectivity
allows you to define the upstream and downstream connections.

Upstream
specifies the upstream heat exchanger group.

Downstream
specifies the downstream heat exchanger group.

Heat Transfer Model
allows you to select either the **simple-effectiveness-model** or the **ntu-model**. See Choosing a Heat Exchanger Model (p. 920) for information on the differences between these models.

Core Porosity Model
specifies whether default values are chosen for the core porosity model.

Edit...
opens the Core Porosity Model Dialog Box (p. 2211) for the definition of a new core porosity model.

Heat Exchanger Performance Data
contains the parameters for heat transfer.

Heat Transfer Data...
opens the Heat Transfer Data Table Dialog Box (p. 2204). This dialog box allows you to define the heat transfer for different primary and auxiliary fluid flow rates. This item will appear for the NTU model only.

Velocity Effectiveness Curve...
opens the Velocity Effectiveness Curve Dialog Box (p. 2210) in which you can define the effectiveness of the heat exchanger core (ε in Equation 6.11 in the Theory Guide). This item will appear for simple effectiveness model only.

Auxiliary Fluid Temperature
specifies the auxiliary fluid temperature. This item will appear for the NTU model only.

Primary Fluid Temperature
specifies the gas stream fluid temperature. This item will appear for the NTU model only.

Geometry
contains parameters to define the macro grid.

Width, Height, Depth
specifies the width, height and the depth of the heat exchanger.

Number of Passes
specifies the number of passes.

Number of Rows/Pass
specifies the number of rows per pass.

Number of Columns/Pass
specifies the number of columns per pass.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

View Passes
displays the macro grid. (This button becomes available after you click **Set**.) The path of the auxiliary fluid is color-coded in the display: macro 0 is red and macro $n-1$ is blue.

Draw Mesh
toggles between displaying and not displaying the mesh when the macro mesh is displayed (using the **View Passes** button). The Mesh Display Dialog Box (p. 2163) opens when **Draw Mesh** is selected.

Update from Plane Tool
updates the **Auxiliary Fluid Inlet Direction** and **Pass-to-Pass Direction** from the plane tool orientation. The **Width**, **Height**, and **Depth** will also be updated. See Using the Plane Tool (p. 1724) for information about using the plane tool.

Auxiliary Fluid Inlet Direction
specifies the direction in which the auxiliary fluid enters the heat exchanger. See Figure 13.17: 1x4x3 Macros (p. 939).

Pass-to-Pass Direction
specifies the direction in which the auxiliary fluid moves at the end of each pass through the heat exchanger. See Figure 13.17: 1x4x3 Macros (p. 939).

Auxiliary Fluid
contains inputs to specify the properties of the auxiliary fluid.

Properties Method
specifies the method to specify the auxiliary fluid properties. You can choose from **constant-specific-heat** and **user-defined-enthalpy**.

Specific Heat
sets the specific heat of the auxiliary fluid (c_p in Equation 6.16 you choose **constant-specific-heat**.

Enthalpy UDF
sets the enthalpy as defined by the user-defined function selected from the drop-down list.

Auxiliary Fluid Flow Rate
sets the flow rate of the auxiliary fluid ($\dot{m}$ in Equation 6.16 in the Theory Guide).

Initial Temperature
sets an initial guess for the inlet temperature (T_{in} in Equation 6.11 and Equation 6.16 in the Theory Guide). This value is specified only if **Fixed Heat Rejection** is selected.

Inlet Pressure
sets the auxiliary fluid inlet pressure. This value is specified only if **user-defined-enthalpy** is selected.

Inlet Quality
specifies the value of x in Equation 6.20 in the Theory Guide.

Pressure Drop
specifies the value of p_{in} in Equation 6.21 in the Theory Guide. This value is specified only if **user-defined-enthalpy** is selected.

Supplementary Auxiliary Fluid Stream
specifies properties of the supplementary auxiliary stream.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Supplementary Mass Flow Rate
specifies the supplementary fluid flow rate as **constant**, **polynomial** or **piecewise-linear**.

Supplementary Flow Temperature
specifies the supplementary fluid temperature as **constant**, **polynomial** or **piecewise-linear**.

Supplementary Flow Quality
specifies the supplementary fluid quality.

Create
saves all the settings in the dialog box.

Delete
deletes the group that is selected in the **HX Groups** list.

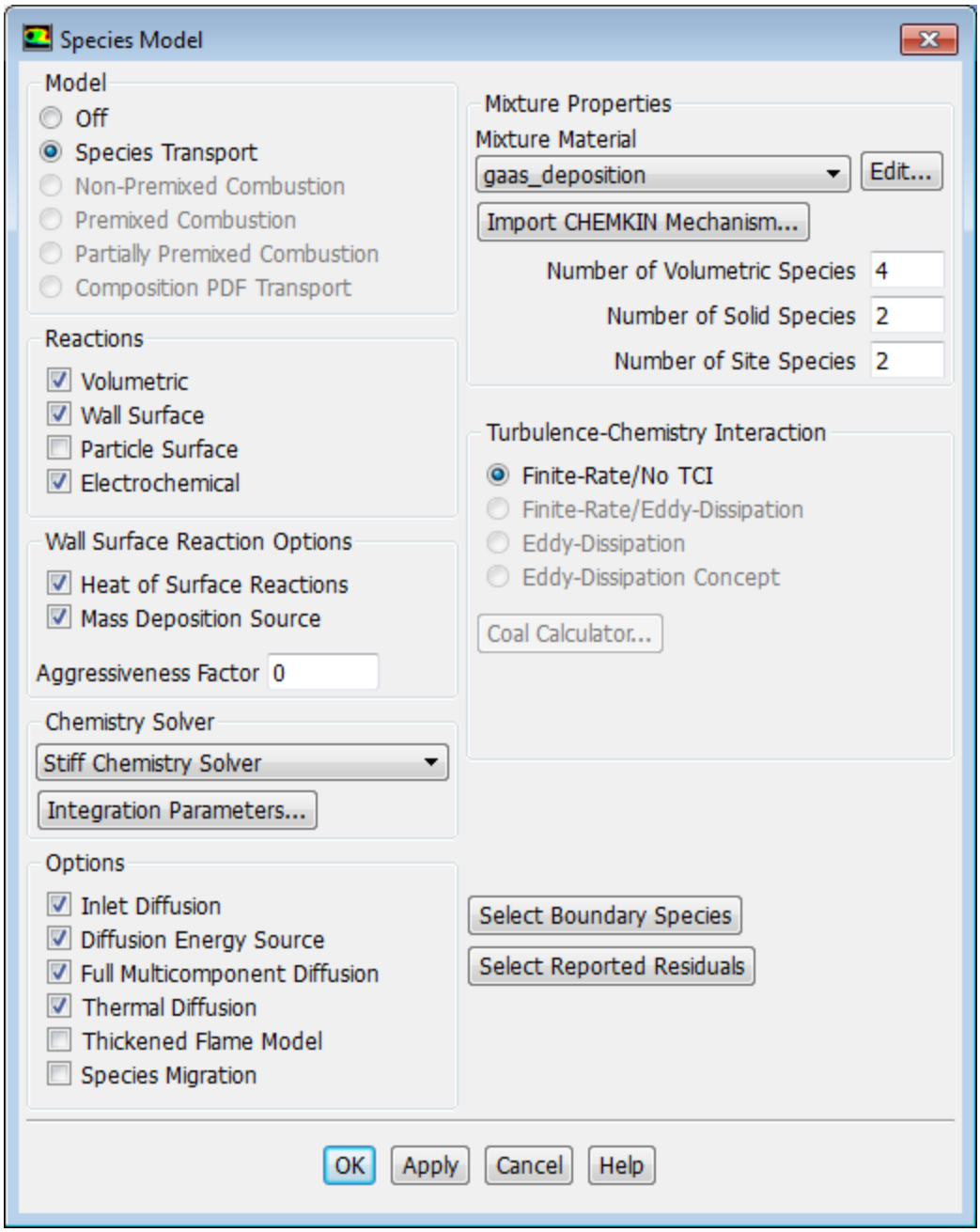
Replace
changes the parameters of the already existing group that is selected in the **HX Groups** list.

Set...
opens the Ungrouped Macro Heat Exchanger Dialog Box (p. 2207), in which you can define a new heat exchanger core model or read one from an external file.

37.4.18. Species Model Dialog Box

The **Species Model** dialog box allows you to set parameters related to the calculation of species transport and combustion. See Enabling Species Transport and Reactions and Choosing the Mixture Material (p. 960), Manual Inputs for Wall Surface Reactions (p. 998), User Inputs for Particle Surface Reactions (p. 1001), Using the Premixed Combustion Model (p. 1090), Using the Partially Premixed Combustion Model (p. 1100), and Steps for Using the Composition PDF Transport Model (p. 1113) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Model

indicates which model, if any, is used to calculate species transport/combustion.

Off

disables species calculations.

Species Transport

enables the calculation of multi-species transport (either non-reacting or reacting, depending on the selection for **Reactions**). See Modeling Species Transport and Finite-Rate Chemistry (p. 957) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Non-Premixed Combustion
enables the calculation of turbulent reacting flow using the non-premixed combustion model. See Modeling Non-Premixed Combustion (p. 1025) for details. This option is available only for turbulent flows using the pressure-based solver.

Premixed Combustion
enables the premixed turbulent combustion model. See Modeling Premixed Combustion (p. 1089) for details. This option is available only for turbulent flows using the pressure-based solver.

Partially Premixed Combustion
enables the partially premixed turbulent combustion model. See Modeling Partially Premixed Combustion (p. 1099) for details. This option is available only for turbulent flows using the pressure-based solver.

Composition PDF Transport
enables the composition PDF transport model. See Modeling a Composition PDF Transport Problem (p. 1113) for details. This option is available only for turbulent flows using the pressure-based solver.

Reactions
contains options related to the modeling of reacting flow. (This section of the dialog box appears only when **Species Transport** or **Composition PDF Transport** is the specified **Model**.)

Volumetric
enables the calculation of reacting flow using the finite-rate formulation. See Volumetric Reactions (p. 958) for details.

Wall Surface
enables the calculation of wall surface reactions. See Wall Surface Reactions and Chemical Vapor Deposition (p. 995) for details. This item will appear only if **Volumetric** is enabled.

Particle Surface
enables the calculation of particle surface reactions. See Particle Surface Reactions (p. 1001) for details. This item will appear only if **Volumetric** is enabled.

Electrochemical
enables the calculation of electrochemical reactions. See Electrochemical Reactions (p. 1003) for details. This item will appear only if **Volumetric** is enabled.

Integration Parameters...
is a command button that opens the Integration Parameters Dialog Box (p. 2235). This button appears for the species transport model, when **Volumetric** is enabled under **Reactions** and **Stiff Chemistry Solver** is enabled under **Options** or when **Eddy-Dissipation Concept** is enabled under **Turbulence-Chemistry Interaction**.

Wall Surface Reaction Options
contains additional options for wall surface reactions. This portion of the dialog box appears only if **Wall Surface** is enabled under **Reactions**.

Heat of Surface Reactions
(if enabled) includes the heat release due to surface reactions in the energy equation. You must remember to set appropriate formation enthalpies (standard state enthalpies) if you enable this option.

Mass Deposition Source
(if enabled) includes the effect of surface mass transfer in the continuity equation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Aggressiveness Factor
is a numerical factor that controls the robustness and the convergence speed. This value ranges between 0 and 1, where 0 is the most robust, but results in the slowest convergence. The default value for the **Aggressiveness Factor** is 0.5.

Chemistry Solver
is a drop-down list in which you can select the solver for the chemically reactive flow simulation. This item appears only for **Species Transport** and **Composition PDF Transport** models involving **Volumetric** reactions.

Stiff Chemistry Solver
enables efficient integration of stiff chemical kinetics. This item appears only when **Finite-Rate/No TCI** or **Eddy-Dissipation Concept** is selected in the **Turbulence-Chemistry Interaction** group box. See Solution of Stiff Chemistry Systems (p. 992) for details.

CHEMKIN-CFD Solver
enables the integration of chemical kinetics using the ANSYS CHEMKIN-CFD Solver, designed for large, stiff chemistry mechanisms. This item appears only when **Finite-Rate/No TCI** or **Eddy-Dissipation Concept** is selected in the **Turbulence-Chemistry Interaction** group box.

Relax to Chemical Equilibrium
enables efficient modeling of chemical kinetics effects based only on equilibrium calculation and assumptions about characteristic timescales for approaching equilibrium. This item is not available for the **Eddy Dissipation Concept TCI** option. (See The Relaxation to Chemical Equilibrium Model for details.

None - Explicit Source
makes explicit use of chemistry source terms in the species transport equations, without a stiff-chemistry solver (not recommended for stiff or complex chemistry).

Options
contains additional options for the **Species Transport** model and for the **Composition PDF Transport**. (This section will not appear in the dialog box for the other models.)

Inlet Diffusion
includes the diffusion flux of species at inlet.

Diffusion Energy Source
(if enabled) includes the effect of enthalpy transport due to species diffusion in the energy equation.

Full Multicomponent Diffusion
enables the full multicomponent diffusion model. See Full Multicomponent Diffusion (p. 520) for details.

Thermal Diffusion
enables the thermal diffusion model. See Thermal Diffusion Coefficients (p. 522) for details.

Liquid Micro-Mixing
is used to model liquid reactions. When the **Liquid Micro-Mixing** model is invoked, ANSYS Fluent uses the **volume-weighted-mixing-law** formula to calculate the density.

Species Migration
(if enabled) includes the ion species migration term in the species transport equation. This item appears only when **Electrochemical** reactions are enabled.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Thickened Flame Model
enables the modeling of laminar flames. This application is typically used as an LES combustion model for turbulent premixed and partially-premixed flames.

Include Temperature Fluctuations
enables the calculation of the multi-mode energy equation. This option is available when the composition PDF transport model is selected.

Mixture Properties
contains controls and information about the mixture being modeled. This section of the dialog box will not appear if **Premixed Combustion** is the selected under **Model**.

Mixture Material
contains a drop-down list of available mixture materials. When you first enable the **Species Transport** model, you can choose from all of the mixture materials defined in the database, or you can choose a "template" and define your own material. (Click **View...** to open the Fluent Database Materials Dialog Box (p. 2312) and check the properties of the mixture material selected in the list.) See Enabling Species Transport and Reactions and Choosing the Mixture Material (p. 960) for details.

When you use the **Non-Premixed Combustion** or **Partially Premixed Combustion** model, this list will be inactive. The mixture material for a non-premixed or partially premixed combustion calculation will be determined from the content of the PDF file generated in ANSYS Fluent using the **PDF Options** parameters.

Number of Volumetric Species
displays the number of gas-phase species in the selected **Mixture Material**. This is an informational display only; you cannot edit this value.

Number of Solid Species
displays the number of solid species defined in the selected **Mixture Material**. This is an informational display only; you cannot edit this value. (This list will appear only for **Species Transport** models involving **Wall Surface** reactions.)

Number of Site Species
displays the number of site species defined in the selected **Mixture Material**. This is an informational display only; you cannot edit this value. (This list will appear only for **Species Transport** models involving **Wall Surface** reactions.)

Turbulence-Chemistry Interaction
indicates which model is to be used for turbulence-chemistry interaction when the **Species Transport** model with **Volumetric** reactions is used.

Laminar Finite-Rate
computes only the Arrhenius rate (see Equation 7.7 in the Theory Guide) and neglects turbulence-chemistry interaction.

Finite-Rate/Eddy-Dissipation
(for turbulent flows) computes both the Arrhenius rate and the mixing rate and uses the smaller of the two.

Eddy-Dissipation
(for turbulent flows) computes only the mixing rate (see Equation 7.25 and Equation 7.26 in the Theory Guide).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Eddy-Dissipation Concept
(for turbulent flows) models turbulence-chemistry interaction with detailed chemical mechanisms (see Equation 7.25 and Equation 7.26 in the Theory Guide).

Coal Calculator...
opens the Coal Calculator Dialog Box (p. 2232).

Select Boundary Species...
opens the **Select Boundary Species** dialog box (see Figure 14.3: The Select Boundary Species Dialog Box (p. 966)).

Select Reported Residuals
opens the **Select Residual Monitored Species** dialog box (see Figure 14.4: The Select Residual Monitored Species (p. 967)).

Options
contains parameters related to the Laminar Finite-Rate or the Eddy-Dissipation Concept model. This section of the dialog box will appear when **Laminar Finite-Rate** or the **Eddy-Dissipation Concept** is selected for **Turbulence-Chemistry Interaction**.

Flow Iterations Per Chemistry Update
specifies how often ANSYS Fluent will update the chemistry during the calculation. Increasing the number can reduce the computational expense of the chemistry calculations.

Aggressiveness Factor
is a numerical factor that controls the robustness and the convergence speed. This value ranges between 0 and 1, where 0 is the most robust, but results in the slowest convergence. The default value for the **Aggressiveness Factor** is 0.5.

Volume Fraction Constant
specifies the value of C_{ξ} in Equation 7.28 in the Theory Guide.

Time Scale Constant
specifies the value of C_{τ} in Equation 7.29 in the Theory Guide.

Number of Grid Points in Flame
by default are 8 grid points.

PDF Options
contains options related for the non-premixed combustion model. (This section will appear only if **Non-Premixed Combustion** or **Partially-Premixed Combustion** is the selected **Model**.)

Inlet Diffusion
includes the diffusion flux of species at inlet.

Compressibility Effects
can be enabled to account for cases where substantial pressure changes occur in time and/or space when modeling a non-adiabatic system. See Specifying the Operating Pressure for the System (p. 1033) for details.

Liquid Micro-Mixing
is used to model liquid reactions. When the **Liquid Micro-Mixing** model is invoked, ANSYS Fluent uses the **volume-weighted-mixing-law** formula to calculate the density.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chemistry
tab contains the parameters to define problems using the chemistry model. See Setting Up the Equilibrium Chemistry Model (p. 1031) for details.

State Relation

Chemical Equilibrium
enables the equilibrium chemistry model. See Setting Up the Equilibrium Chemistry Model (p. 1031) for details.

Steady Diffusion Flamelet
enables the steady diffusion flamelet model. See The Diffusion Flamelet Models Theory in the Theory Guide for details.

Unsteady Diffusion Flamelet
enables the Eulerian unsteady diffusion flamelet model.

Diesel Unsteady Flamelet
enables the diesel unsteady laminar flamelet model. See Using the Diesel Unsteady Laminar Flamelet Model (p. 1038) for details.

Flamelet Generated Manifold
enables the flamelet generated manifold (FGM) model. This is available when the **Partially Premixed Combustion** model is selected. See Partially Premixed Combustion in the Theory Guide for details.

Energy Treatment

Adiabatic
enables adiabatic modeling options for the problem.

Non-Adiabatic
enables non-adiabatic modeling options for the problem. See Non-Adiabatic Extensions of the Non-Premixed Model in the Theory Guide for details.

Coal Calculator
opens the Coal Calculator Dialog Box (p. 2232).

Stream Options
contains the parameters for the equilibrium chemistry model or the steady diffusion flamelet model.

Secondary Stream
includes the secondary inlet stream in the model.

Empirical Fuel Stream
enables parameters to define fuel stream empirically. This option is available only with the full equilibrium chemistry model.

Empirical Secondary Stream
enables parameters to define secondary stream empirically. This option is available only with the full equilibrium chemistry model.

Model Settings
contains a list of parameter settings.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Operating Pressure
specifies the system operating pressure used to calculate the density using the ideal gas law. See Specifying the Operating Pressure for the System (p. 1033) for details.

Fuel Stream Rich Flamability Limit
specifies the rich flammability limit for fuel stream when the equilibrium chemistry option is used. You will not set these if you have used the empirical definition option for fuel composition. See Enabling the Rich Flammability Limit (RFL) Option (p. 1035) for details.

Secondary Stream Flamability Limit
specifies the rich flammability limit for secondary stream when the equilibrium chemistry option is used. You will not set these if you have used the empirical definition option for fuel composition. See Enabling the Rich Flammability Limit (RFL) Option (p. 1035) for details.

Empirical Fuel Lower Calorific Value
specifies the lower calorific value of fuel stream.

Empirical Fuel Specific Heat
specifies the specific heat value of fuel stream.

Empirical Fuel Molecular Weight
specifies the molecular weight of the fuel stream.

Empirical Secondary Lower Calorific Value
specifies the lower calorific value of secondary stream.

Empirical Secondary Specific Heat
specifies the specific heat value of secondary stream.

Empirical Secondary Molecular Weight
specifies the molecular weight of the secondary stream.

Options
contains options related to the steady flamelet model.

Create Flamelet
enables the **Import CHEMKIN Mechanism...** button that opens the CHEMKIN Mechanism Import Dialog Box (p. 2642) where you can import the CHEMKIN mechanism and thermodynamic data, to create a flamelet file. This option is available for the steady flamelet model. See Setting Up the Steady and Unsteady Diffusion Flamelet Models (p. 1036) for details.

Import Flamelet
enables the **Import Flamelet File...** button that opens The Select File Dialog Box (p. 19) where you can select the existing flamelet in ANSYS Fluent. You can also set the file type parameters to import the existing flamelet in ANSYS Fluent. See Setting Up the Steady and Unsteady Diffusion Flamelet Models (p. 1036) for details. This option is available for the steady flamelet model.

Flamelet Type
gives you the option of creating a **Premixed Flamelet** or a **Diffusion Flamelet**. This is available when the **Flamelet Generated Manifold** model is selected.

File Type
contains the toggle buttons for two flamelet file types.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Standard
enables the import of an ASCII format standard flamelet file.

CFX-RIF
enables the import of an ASCII format CFX-RIF flamelet file.

Flamelet Property File Name
opens The Select File Dialog Box (p. 19) in which you can save the existing flamelet in ANSYS Fluent to use when running an existing case.

Thermodynamic Database File Name
specifies a path for the thermodynamic database file to be read.

Boundary
tab contains the list of boundary species and related parameters. This is available only for equilibrium chemistry model. See Defining the Stream Compositions (p. 1042) for details.

Species
contains the list of the species used in the problem.

Fuel
specifies the fuel species.

Oxid
specifies the oxidizing species.

Second
specifies the secondary species.

Boundary Species
allows you to specify the species you want to add or remove from the model. You can type the species chemical formula in the field below it.

Add
adds the species in the model.

Remove
removes the species from the model.

List Available Species
prints a list of all species in the thermodynamic database file (`thermo.db`) in the console window.

Temperature
specifies the temperature of different streams that you have defined.

Fuel
is the temperature of the fuel inlet in the model.

Oxid
is the temperature of the oxidizer inlet in the model.

Second
is the temperature of the secondary stream inlet in the model.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specify Species in
allows you to define the unit of species concentration.

Mass Fraction
allows you to specify the species in terms of mass fraction.

Mole Fraction
allows you to specify the species in terms of mole fraction.

Control
tab contains the parameters for exclusion and inclusion of equilibrium species. This is available only for equilibrium chemistry model. See Forcing the Exclusion and Inclusion of Equilibrium Species (p. 1053) for details.

Species Excluded from Equilibrium
lists the species excluded from equilibrium calculation.

Species
lists the slow-forming species that are zeroed in the initial flamelet profile.

Add
allows you to add equilibrium species.

Remove
allows you to remove equilibrium species.

List Available Species
prints a list of all species in the thermodynamic database file in the console window.

Flamelet Controls
allows you to adjust the controls for the flamelet solution. Note that the **Create Flamelet** option in the **Chemistry** tab must be selected for the **Steady Diffusion Flamelet** or **Flamelet Generated Manifold** models for these controls to be available.

Initial Fourier Number
sets the first time step for the solution.

Fourier Number Multiplier
increases the time step at subsequent times. Every time step after the first is multiplied by this value.

Relative Error Tolerance and Absolute Error Tolerance
specifies the local error controls during numerical integration.

Flamelet Convergence Tolerance
specifies the maximum absolute change in species fraction or temperature at any discrete mixture-fraction.

Maximum Integration Time
specifies the maximum total elapsed time for flamelet calculation. ANSYS Fluent will stop the flamelet calculation after the total elapsed time has exceeded this value.

Flamelet
tab allows you to adjust the controls for the flamelet solution. See Defining the Flamelet Controls (p. 1054) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Flamelet Parameters

consist of the controls for the flamelet solution.

> **Note**
>
> The parameters may vary slightly if you selected the **Flamelet Generated Manifold** model (see Flamelet Generated Manifold (p. 1103) for those specific parameters).

Number of Grid Points in Flamelet

specifies the number of mixture fraction grid points distributed between the oxidizer ($f=0$) and the fuel ($f=1$).

Maximum Number of Flamelets

specifies the maximum number of laminar flamelet profiles to be calculated.

Initial Scalar Dissipation

is the scalar dissipation of the first flamelet in the library.

Scalar Dissipation Step

specifies the interval between scalar dissipation values (in s^{-1}) for which multiple flamelets will be calculated.

User Defined Flamelet Parameters

enables you to hook a user-defined function for scalar dissipation and mean mixture fraction (or progress variable) grid discretization

Include Equilibrium Flamelet

determines whether the equilibrium assumption is used to compute the progress variable (Equation 9.2 in the *Fluent Theory Guide*). By default, this option is selected. If you turn off this option, ANSYS Fluent will compute the progress variable using the solution of flamelet generated with initial scalar dissipation as a denominator in Equation 9.2 in the *Fluent Theory Guide*.

Unsteady Flamelet Parameters

consist of the controls for the unsteady flamelet solution.

Number of Grid Points in Flamelet

specifies the number of mixture fraction grid points distributed between the oxidizer ($f=0$) and the fuel ($f=1$).

Mixture Fraction Lower Limit for Initial Probability

is the limit at which the unsteady flamelet model temporally convects and diffuses a marker probability equation through a steady-state ANSYS Fluent flow-field.

Maximum Scalar Dissipation

is where flamelets extinguish at large scalar dissipation (mixing) rates.

Courant Number

is the number at which ANSYS Fluent automatically selects the time step for the probability equation based on this convective Courant number.

Number of Flamelets

specifies the number of unsteady laminar flamelets that ANSYS Fluent will automatically generate during the simulation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculate Flamelets
begins the laminar flamelet calculation.

Display Flamelets...
opens the Flamelet 3D Surfaces Dialog Box (p. 2237) from which you can display 2D plots and 3D surfaces showing the variation of species fraction or temperature with the mean mixture fraction or scalar dissipation.

Initialize Unsteady Flamelet Probability
initializes the unsteady flamelet and its probability marker equation.

Display Unsteady Flamelet...
opens the Flamelet 2D Curves Dialog Box (p. 2239) from which you can display 2D plots of the different variables.

Table
tab contains parameters to create the look-up table. See Calculating the Look-Up Tables (p. 1063) for details of the items listed below.

Note

If you selected the **Flamelet Generated Manifold** model, the parameters will be different. See Calculating the Look-Up Tables (p. 1105) for those specific parameters.

Table Parameters
consists of the controls for the lookup table. A different set of parameters for you to enter will be displayed if **Automated Grid Refinement** is enabled or disabled.

Initial Number of Grid Points
specifies the number of grid points for the resolution of the mean mixture fraction, mixture fraction variance, and mean enthalpy (for non-adiabatic systems).

Maximum Number of Grid Points
specifies the maximum number of grid points used for tabulation. The grid refinement procedure will stop inserting the points when either the change in value and slope between successive points is within tolerance or the maximum number of grid points are generated.

Maximum Change in Value Ratio
specifies the maximum allowable change in value of table variables between successive grid points as specified by Equation 8.27 in the Theory Guide.

Maximum Change in Slope Ratio
specifies the maximum change in the slope of table variables between successive grid points as specified by Equation 8.28 in the Theory Guide.

Maximum Number of Species
is the maximum number of species stored in the lookup tables.

Minimum Temperature
specifies the minimum temperature in the lookup tables.

Number of Mean Mixture Fraction Points
is the number of discrete values of $\overline{f}$ at which the look-up tables will be computed.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number of Secondary Mixture Fraction Points
is the number of discrete values of p_{sec} at which the look-up tables will be computed. This option is available only when a secondary stream has been defined.

Number of Mixture Fraction Variance Points
is the number of discrete values of $\overline{f'^2}$ at which the look-up tables will be computed. This option is available only when no secondary stream has been defined.

Maximum Number of Species
is the maximum number of species that will be included in the look-up tables.

Number of Mean Enthalpy Points
is the number of discrete values of enthalpy at which the three-dimensional look-up tables will be computed. This input is required only if you are modeling a non-adiabatic system.

Minimum Temperature
is used to determine the lowest temperature for which the look-up tables are generated (see Figure 8.10: Visual Representation of a Look-Up Table for the Scalar as a Function of Mean Mixture Fraction and Mixture Fraction Variance and Normalized Heat Loss/Gain in Non-Adiabatic Single-Mixture-Fraction Systems in the Theory Guide). This option is available only if you are modeling a non-adiabatic system.

Automated Grid Refinement
is an adaptive algorithm that inserts grid points in all table dimensions so that changes in the values of tabulated variables (such as mean temperature, density and species mass fractions) between successive grid points, as well as changes in their slopes, are less than a user specified tolerance.

Include Equilibrium Flamelet
specifies that an equilibrium flamelet (that is, $\chi=0$) will be generated in ANSYS Fluent and appended to the flamelet library before the PDF table is calculated. This option is available only when you are generating more than one laminar flamelet.

Calculate PDF Table
generates the look-up table.

Display PDF Table
opens the PDF Table Dialog Box (p. 2240) where you can display 2D plots and 3D surfaces showing the variation of species mole fraction, density, or temperature with the mean mixture fraction, mixture fraction variance, or enthalpy.

Properties
tab contains parameters needed to modify the piecewise-linear points. See Modifying the Unburnt Mixture Property Polynomials (p. 1110) for the details.

Partially Premixed Mixture Properties
contains the list of properties that you can modify. **Edit...** opens the **Quadratic Mixture Fraction** dialog box where you can modify the values of the polynomial coefficients.

For each polynomial function of $\overline{f}$ under **Partially Premixed Mixture Properties** you can click **Edit...** and specify values for **Coefficient 1**, **Coefficient 2**, **Coefficient 3**, and **Coefficient 4** in the appropriate **Quadratic of Mixture Fraction** dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Non-Adiabatic Laminar Flame Speed
when enabled includes the non-adiabatic effects on the laminar flame speed by tabulating the laminar speeds in the PDF table. See Laminar Flame Speed in the Theory Guide.

Recalculate Properties
will calculate the partially premixed properties.

Premix
tab contains the options and the parameters for the **Turbulent Flame Speed Model**, the **Variance Settings**, and the **Turbulence-Chemistry Interaction**.

Turbulent Flame Speed Model
contains the **Flame Speed Model** drop-down list, which allows you to choose between the **zimont** or the **peters** model constants for the Zimont premixed combustion model. (This section will appear only if **Premixed Combustion** or **Partially Premixed Combustion** is the selected **Model** and if the **C Equation** or **G Equation** premixed model is chosen.)

Turbulent Length Scale Constant
specifies the value of C_D in Equation 9.11 in the Theory Guide.

Turbulent Flame Speed Constant
specifies the value of A in Equation 9.9 in the Theory Guide.

Stretch Factor Coefficient
specifies the value of μ_{str} in Equation 9.16 in the Theory Guide.

Turbulent Schmidt Number
specifies the value of Sc_t in Equation 9.1 in the Theory Guide.

Wall Damping Coefficient
specifies the value of α_w in Equation 9.19.

Ewald Corrector
is enabled by default and described in Peters Flame Speed Model.

G Equation Settings
allows you to select either the **transport equation** or **algebraic** option for the calculation of the flame distance variance. Consult Peters Flame Speed Model in the Theory guide for the variance transport and algebraic equation expressions (Equation 9.6 and Equation 9.7).

Flame Curvature Source
includes the curvature source term in the G-Equation, which is the last term in Equation 9.4.

Turbulence-Chemistry Interaction
is available if you select **Flamelet Generated Manifold** for the partially premixed combustion model. The three available options (**Finite-Rate**, **Turbulent Flame Speed**, and **Finite-Rate/Turbulent Flame Speed**) are discussed in FGM Turbulent Closure in the Theory Guide.

Variance Settings
is available for the **G Equation** and **C Equation**. (This section will appear only if **Premixed Combustion** or **Partially Premixed Combustion** is the selected **Model**.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Variance Method
contains a drop-down list of the available two options: **transport equation** and **algebraic**. Consult Peters Flame Speed Model in the Theory Guide for the variance transport and algebraic equation expressions (Equation 9.6 and Equation 9.7). It is recommended that you use the **transport equation** option for RANS and the **algebraic** option for LES.

Premixed Combustion Model Options
contains options for the premixed combustion model. (This section will appear only if **Premixed Combustion** is the selected **Model**.)

Adiabatic
enables the adiabatic premixed combustion model, which calculates temperature using Equation 9.65 in the Theory Guide.

Non-Adiabatic
enables the non-adiabatic premixed combustion model, which calculates temperature using Equation 9.66 in the Theory Guide.

Premixed Model
contains options for choosing a premixed model.

C Equation
allows you to choose the C equation as described in C-Equation Model Theory.

Extended Coherent Flamelet Model
allows you to choose the Extended Coherent Flamelet model as described in Extended Coherent Flamelet Model Theory.

G Equation
allows you to choose the G equation as described in G-Equation Model Theory.

Extended Coherent Flamelet Model Constants
contains model constants for the Extended Coherent Flame Model. (This section will appear only if **Premixed Combustion** or **Partially Premixed Combustion** is the selected **Model** and if the **Extended Coherent Flame Model** flame speed model is chosen.) See Modifying the Constants for the ECFM Flame Speed Closure (p. 1095) for details.

ITNFS Treatment
contains a drop-down list of the available ITNFS treatments: **constant-delta**, **meneveau**, **blint**, **poinsot**, and **constant**.

ITNFS Flame Thickness
sets the flame thickness (Equation 9.29 in the Theory Guide).

Turbulent Schmidt Number
set the turbulent Schmidt number (Sc_t).

Wall Flux Coefficient
set the wall flux coefficient.

PDF Transport Options
contains options for the Composition PDF Transport combustion model. (This section will appear only if **Composition PDF Transport** is the selected **Model**.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lagrangian
solves the composition PDF transport equation by stochastically tracking Lagrangian particles through the domain.

Eulerian
assumes a shape for the PDF, allowing Eulerian transport equations to be derived.

Mixing
tab contains the mixing models.

Mixing Model
contains options to specify the method for modeling molecular diffusion. (This section will appear only if **Composition PDF Transport** is the selected **Model**.) See Particle Mixing in the Theory Guide for details.

Modified Curl
enables the modified curl model for molecular diffusion.

IEM
enables the IEM model for molecular diffusion.

EMST
enables the EMST mixing model for molecular diffusion.

Mixing Constant
specifies the value of the mixing constant C_φ in Equation 11.6 and Equation 11.8 in the Theory Guide.

Boundary
tab allows you to define the fuel and oxidizer compositions. This is only available if you select **Eulerian** as the **PDF Transport Option**.

Species
consists of the fuel species and the oxidizer.

Fuel
is the mole or mass fraction of the fuel stream. The sum of mass or mole fractions of all species in the fuel stream should be 1.

Oxidizer
is the mole or mass fraction of the oxidizer stream. The sum of mass or mole fractions of all species in the fuel oxidizer stream should be 1.

Specify Species in
specifies the species as a **Mass Fraction** or **Mole Fraction**.

Control
tab contains Lagrangian PDF transport parameters.

PDF Transport Parameters
allows you to set the **Particles Per Cell**.

Particles Per Cell
sets the number of PDF particles per cell. Higher values of this parameter will reduce statistical error, but increase computational time.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Local Time Stepping
toggles the calculation of local time steps. If this option is disabled, then you will need to specify the **Time Step** directly (see Equation 11.4 in the Theory Guide). This option is available for steady-state simulations.

Convection #
specifies the particle convection number (see Δt_{conv} in Equation 11.4 in the Theory Guide).

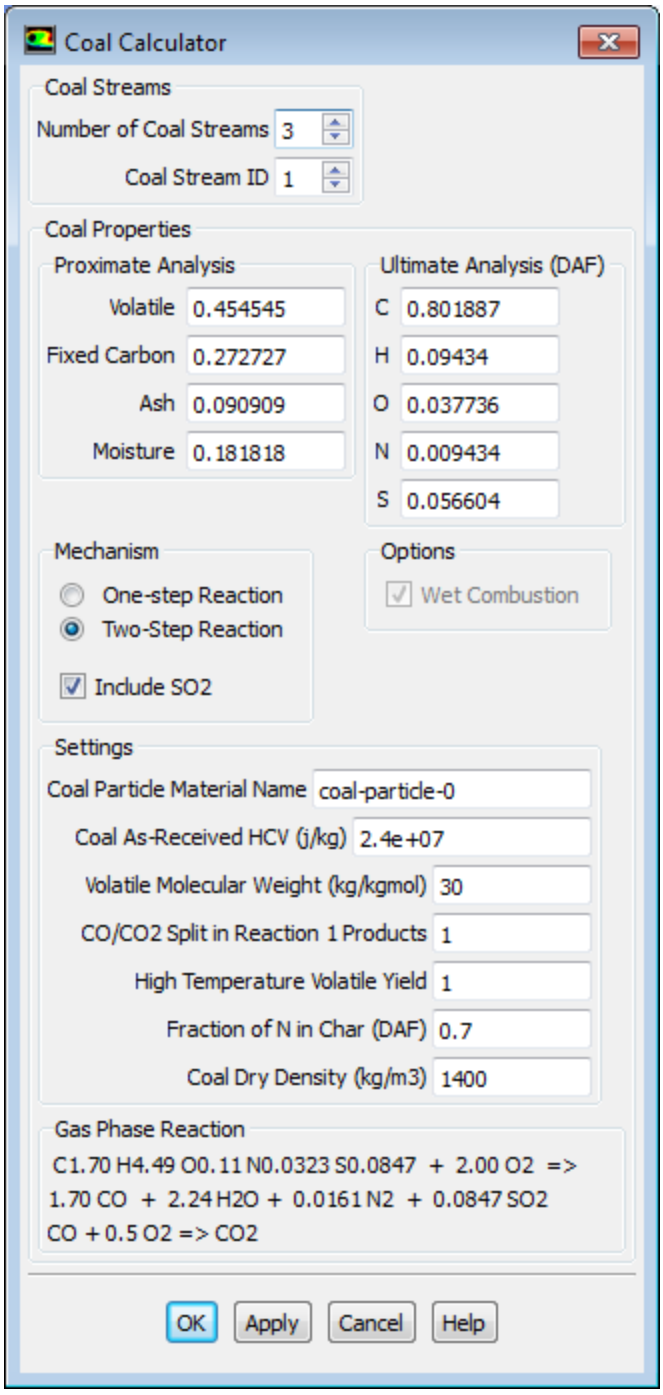
Diffusion #
specifies the particle diffusion number (see Δt_{diff} in Equation 11.4 in the Theory Guide).

Mixing #
specifies the particle mixing number (see Δt_{mix} in Equation 11.4 in the Theory Guide).

37.4.19. Coal Calculator Dialog Box

The **Coal Calculator** dialog box (opened by clicking **Coal Calculator...** in the **Species Model** dialog box) automates the calculations described in Additional Coal Modeling Inputs in ANSYS Fluent (p. 1049).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Coal Streams

allows you to set up to three coal streams (Species Transport model only).

Number of Coal Streams

specifies the total number of coal streams.

ID

sets the ID of the coal stream for which you want to specify **Coal Properties**.

Coal Properties

contains inputs for the current coal stream.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Proximate Analysis
is the mass fraction of **Volatile**, **Fixed Carbon**, **Ash**, and **Moisture** in the coal.

Volatile
is the fraction of the volatile component.

Fixed Carbon
is calculated as one minus the sum of the actual **Volatile**, **Ash**, and **Moisture** fractions.

Ash
is the fraction of ash.

Moisture
is the moisture fraction in the coal.

Ultimate Analysis (DAF)
is the mass fraction of atomic **C, H, O, N** and optionally **S**, in the Dry-Ash-Free (DAF) coal.

Mechanism
allows you to set the mechanisms.

Secondary Stream
when enabled, allows you to set the two mixture fraction model with the primary stream representing char as C <s> , and an empirical secondary stream representing the volatiles. This is available when using the non-premixed combustion model.

One-step Reaction
is defined in Equation 14.6 (p. 988).

Two-step Reaction
involves oxidation of volatiles to CO in the first reaction and oxidation of CO to CO_2 in the second reaction, as described in Equation 14.7 (p. 988).

Include SO2
when enabled, allows you to specify the atomic mass fraction of sulphur, **S**, which appears under **Ultimate Analysis**.

Options

Wet Combustion
when enabled will enable the DPM **Wet Combustion** option by default in all injections created after the **OK** button is clicked in the **Coal Calculator** dialog box.

Settings
is where you will specify the values used in the calculation.

Coal Particle Material Name
is the name of the DPM combusting particle material. The default name is *coal-particle*.

Coal As-Received HCV
is the higher caloric value of the coal.

Volatile Molecular Weight
is the molecular weight of pure volatiles.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CO/CO2 Split in Reaction 1 Products
can be used to specify the molar fraction of CO to CO_2 in the first reaction of Equation 14.7 (p. 988). The default value of 1 implies that all carbon is reacted to CO, with no CO_2 produced.

High Temperature Volatile Yield
is where the enhanced devolatization at higher temperatures can cause the volatile yield to exceed the proximate analysis fraction.

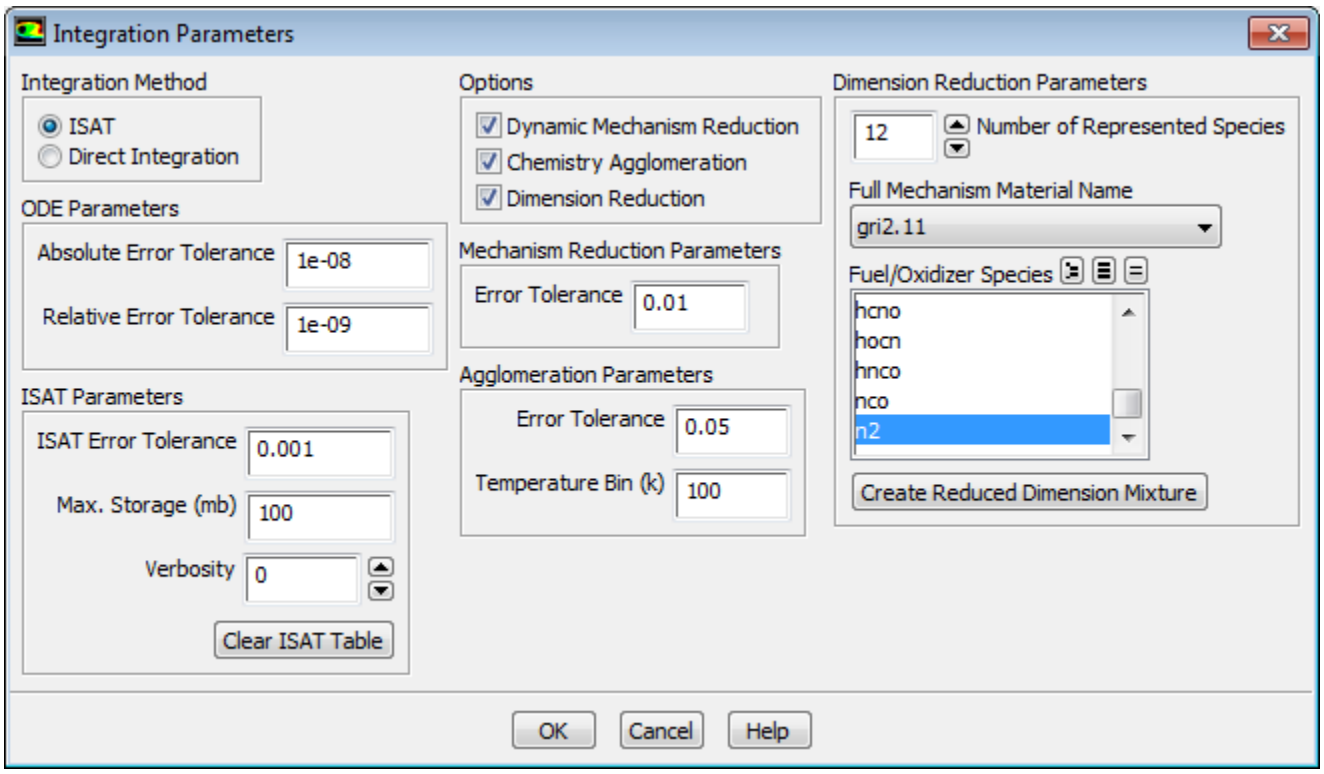
Fraction of N in Char (DAF)
is used in calculating the split of atomic nitrogen for the Fuel NOx model.

Coal Dry Density
is used to calculate the **Volume Fraction** of liquid-water for the **Wet Combustion** option in the **Injections** dialog box.

Gas Phase Reaction
lists the reaction based on your entries for the proximate and ultimate analyses.

37.4.20. Integration Parameters Dialog Box

The **Integration Parameters** dialog box allows you to set the parameters for the integration of the chemical source term S in Equation 11.10 in the Theory Guide. See Using ISAT (p. 1128) for details.



Integration Method
contains options to choose the method for integration.

ISAT
enables the ISAT option and expands the dialog box to include inputs for **ISAT Parameters**.

Direct Integration
enables the direct integration method to integrate the chemical source term in the calculation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ODE Parameters
contains options to specify the error tolerances.

Absolute Error Tolerance
specifies the absolute error tolerance.

Relative Error Tolerance
specifies the relative error tolerance.

ISAT Parameters
contains inputs required for ISAT integration method.

ISAT Error Tolerance
controls the numerical error in ISAT liner interpolation. Decrease this value to get accurate minor species and pollutant predictions.

Max. Storage
is the maximum RAM used by the ISAT table, and has a default value is 100 MB.

Verbosity
specifies the level of detail at which you can monitor the ISAT performance.

Clear ISAT Table
purges the ISAT table.

Options
contains options to choose the method for chemistry acceleration.

Dynamic Mechanism Reduction
accelerates chemistry by reducing the chemical kinetics mechanism on-the-fly to include only important species and reactions, with a corresponding decrease in accuracy. This method is available for **Stiff Chemistry Solver** only.

Chemistry Agglomeration
when enabled, provides additional run-time improvement, with a corresponding decrease in accuracy. This method is available for **Stiff Chemistry Solver** and **Relax to Chemical Equilibrium** only.

Dimension Reduction
is a chemistry acceleration method in addition to ISAT storage-retrieval and Cell Agglomeration, providing faster chemistry calculations with a corresponding loss of accuracy. This method is available for **Stiff Chemistry Solver** only.

Dynamic Cell Clustering
(transient flow) enables a computationally efficient dynamic cell clustering method (DCC) that groups computational cells of high similarity into clusters using two DCC parameters:
- **Max. Temperature Dispersion** (default =10 K)
- **Max. Equiv. Ratio Dispersion** (default =0.05)

This option is enabled by default and recommended. It is available only with the **CHEMKIN-CFD Solver**.

Agglomeration Parameters
contains settings for the agglomeration chemistry method.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Error Tolerance
determines the size of the clusters for species. By default, the value is 0.05.

Temperature Bin
specifies the maximum bin size for temperature. The default value is 100 K.

Dimension Reduction Parameters
contains settings to accelerate the chemistry.

Number of Represented Species
must be greater than 10 and less than the number of species in the full mechanism. The **Number of Represented Species** must also be less than 50 minus the number of unrepresented elements (the number of chemical elements in the unrepresented species).

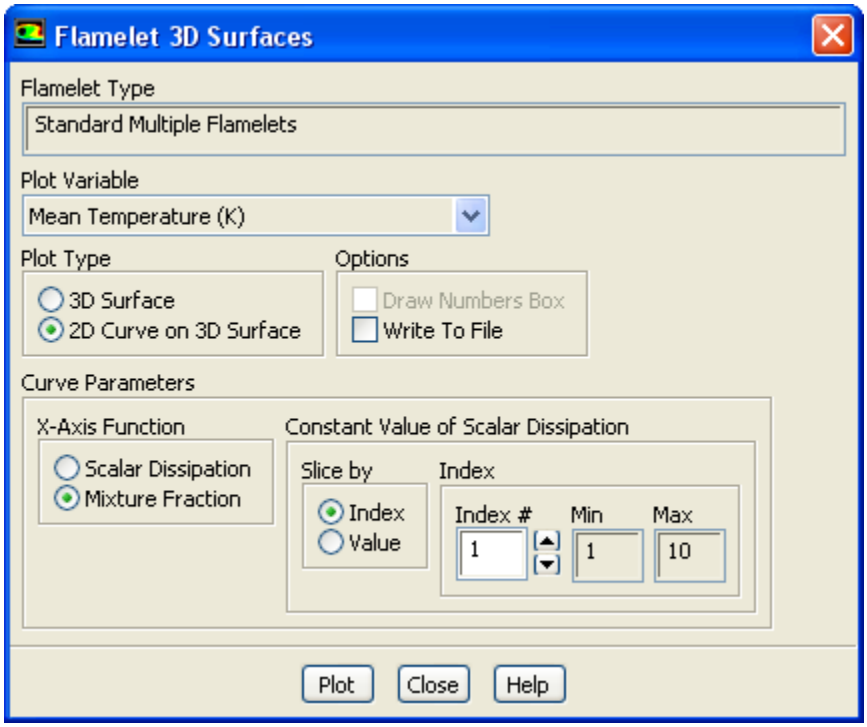
Full Mechanism Material Name
is typically the name of the CHEMKIN mechanism that you imported.

Fuel/Oxidizer Species
is where the boundary and initial fuel and oxidizer, as well as product species, are set as represented species.

Create Reduced Dimension Mixture
creates a new mixture material called `reduced-dimension-mixture`, which contains the represented species as well as proxy 'species' for the unrepresented elements.

37.4.21. Flamelet 3D Surfaces Dialog Box

The **Flamelet 3D Surfaces** dialog box allows you to display 2D plots and 3D surfaces showing the variation of species fraction or temperature with the mean mixture fraction or scalar dissipation. See Postprocessing the Flamelet Data (p. 1060) for details.



Flamelet Type
indicates the type of flamelets whose surfaces will be displayed.

Plot Variable
enables you to choose temperature or species fraction as the variable to be plotted.

Plot Type
consists of options for plot type.

3D Surface
enables plotting on 3D surfaces.

2D curve on 3D surface
enables plotting of a 2D curve on a 3D surface.

Options
consists of the following parameters:

Draw Numbers Box
enables the display of a wireframe box with the numerical limits in each coordinate direction.

Write To File
enables saving the plot data to a file.

Curve Parameters
consists of controls related to plot display.

X-Axis Function
consists of the function against which the plot variable will be displayed.

Scalar Dissipation
enables display of plot variable against the scalar dissipation function.

Mixture Fraction
enables display of plot variable against the mixture fraction.

Constant Value of Scalar Dissipation
consists of the controls to specify the type of discretization (that is, how the flamelet data will be sliced) for the variable that is being held constant.

Slice by
consists of the controls to specify discretization.

Index
enables you to specify discretization index of the variable that is being held constant.

Value
enables you to specify the numerical value of the variable that is being held constant.

Index
consists of the controls that are displayed when you enable **Index** under **Slice by**.

Index #
displays the index number.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Min
displays the minimum of the range of integer values that you are allowed to choose from.

Max
displays the maximum of the range of integer values that you are allowed to choose from.

Value
consists of the controls that are displayed when you enable **Value** under **Slice by**.

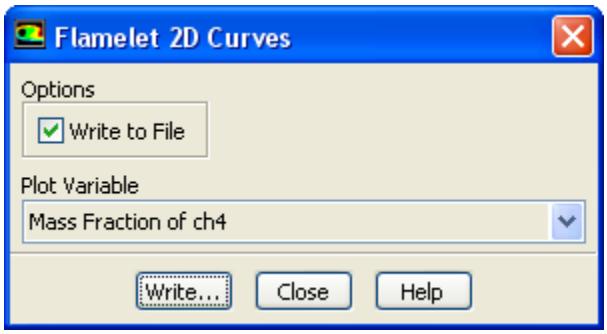
Value
enables you to enter the numerical value of the variable that is being held constant.

Min
displays the minimum of the range of integer values that you are allowed to choose from.

Max
displays the minimum of the range of integer values that you are allowed to choose from.

37.4.22. Flamelet 2D Curves Dialog Box

The **Flamelet 2D Curves** dialog box allows you to display or write 2D curves of the unsteady flamelet. See Postprocessing the Flamelet Data (p. 1060) for details.



Options
gives you the option to plot or **Write to File** 2D curves.

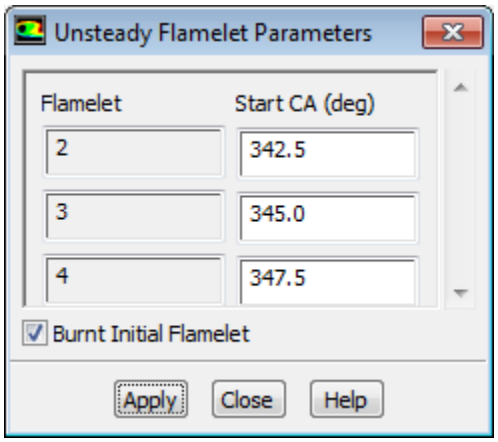
Plot Variable
consists of a drop-down list of variables that you can plot or write.

Plot
is available by default. When the **Write to File** option is enabled the **Plot** button changes to a **Write...**.

37.4.23. Unsteady Flamelet Parameters Dialog Box

The **Unsteady Flamelet Parameters** dialog box (opened by clicking **Set Flamelet Parameters** in the **Flamelet** tab of the **Species Model** dialog box) allows you to specify the initiation time of the unsteady flamelets. See Using the Diesel Unsteady Laminar Flamelet Model (p. 1038) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

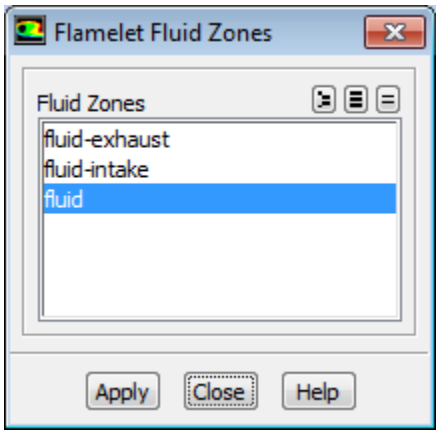


Start CA (deg)
is the time at which each unsteady laminar flamelet will be generated during your simulation. The time is specified in terms of seconds or degrees of crank angle if the dynamic mesh is enabled.

Burnt Initial Flamelet
if enabled, sets the initial flamelet condition to a chemical equilibrium burnt state. Otherwise, the initial flamelet condition is set to unburnt state (default).

37.4.24. Flamelet Fluid Zones Dialog Box

The **Flamelet Fluid Zones** dialog box allows you to select the fluid zones for computing zone-averaged pressure and scalar dissipation. The dialog box is opened by clicking **Set Flamelet Fluid Zones** in the **Flamelet** tab of the **Species Model** dialog box. See Using the Diesel Unsteady Laminar Flamelet Model (p. 1038) for details.



Fluid Zones
is a selectable list of the fluid zones over which the average pressure and scalar dissipation are computed.

37.4.25. PDF Table Dialog Box

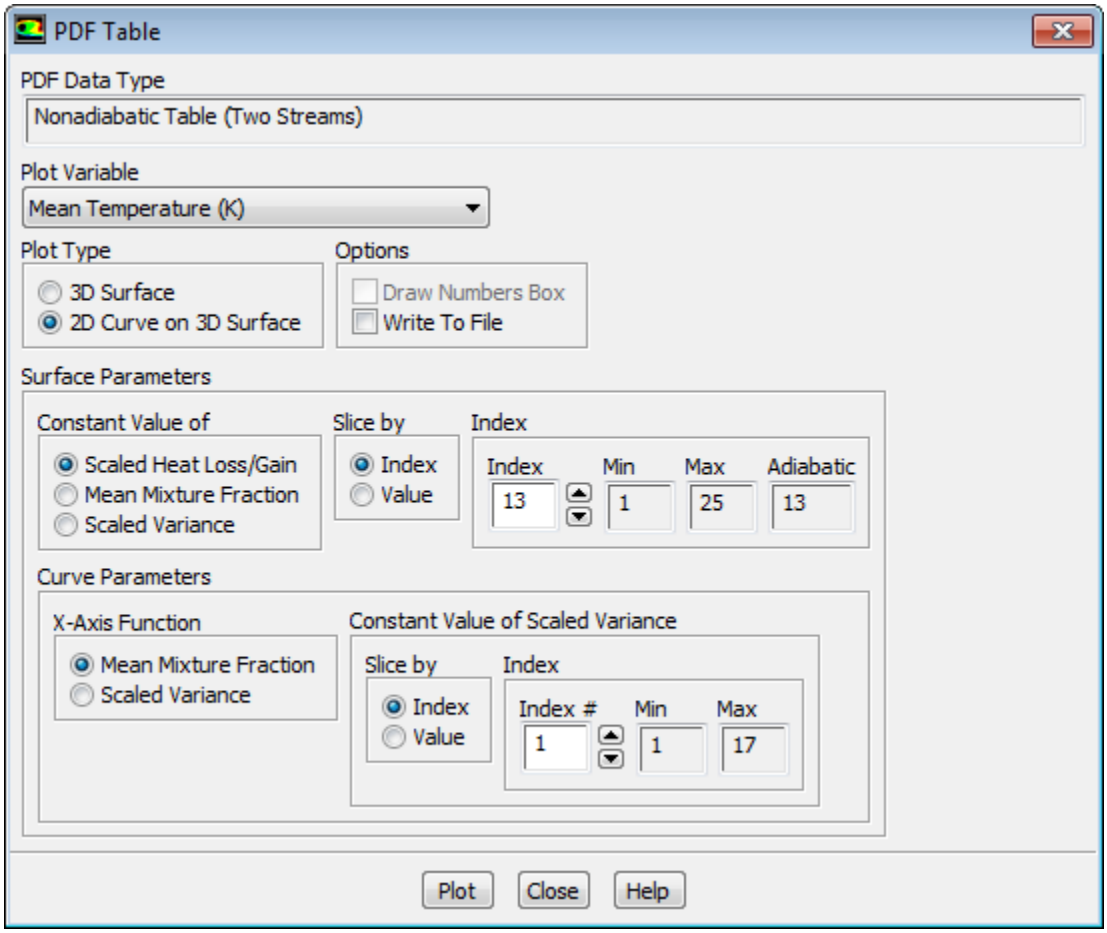
You can display 2D plots and 3D surfaces showing the variation of species mole fraction, density, or temperature with the mean mixture fraction, mixture fraction variance, or enthalpy. For the Flamelet Generated Manifold (FGM) model, additional variables are available for generating plots for PDF lookup tables. See Postprocessing the Look-Up Table Data (p. 1068) for details about the items below.

The **PDF Table** dialog box can be accessed in one of two ways:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- by clicking the **Display PDF Table...** button in the **Table** tab of the **Species Model** dialog box (as shown in Figure 15.23: The Species Model Dialog Box (Table) Tab Excluding Automated Grid Refinement (p. 1065))
- by using the following path:

  Postprocessing → **Model Specific** → **PDF Table...**



Controls

PDF Data Type

describes the system that you are displaying.

Plot Variable

contains a drop-down list from which you can select temperature, density, or species mole fraction as the variable to be plotted. For the FMG model, you can also select mean rate of reaction progress.

Enthalpy Slice, Adiabatic = *slice number*

(FGM model only) specifies the value of the enthalpy slice level. By default, the enthalpy level is fixed at the adiabatic level displayed in the label as **Adiabatic = *slice number***. Enthalpy slice numbers below the adiabatic slice number are associated with heat loss, while enthalpy slice numbers above the adiabatic slice number are associated with enthalpy gain.

Scalar Dissipation

(multiple flamelets only) specifies the value of the **Scalar Dissipation**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plot Type
gives you the choice of plotting a 3D surface or a slice of a 3D surface.

3D Surface
displays a 3D plot of the variation of species mole fraction, density, or temperature with the mean mixture fraction, mixture fraction variance, or enthalpy.

2D Curve on 3D Surface
consists of a 2D curve that is a slice of a 3D surface.

Options
contains options specific to the display of 3D surfaces or 2D curves on 3D surfaces.

Draw Numbers Box
enabling this option displays a wireframe box with the numerical limits in each coordinate direction. This option is available only when **3D Surface** is selected.

Write To File
specifies whether you want to write the plot data to a file. This option is available only when **2D Curve on 3D Surface** is selected. The **Plot** button changes to a **Write** button when this option is enabled.

Volume Parameters
(FGM model only) contains settings for selecting and fixing a discrete independent variable.

Constant Value of
contains the discrete independent variables that can be held constant in the lookup table. The choices are **Mean Reaction Progress**, **Scaled Reaction Progress Variance**, **Mean Mixture Fraction**, and **Scaled Variance**.

Slice by
allows you to select whether the 3D array of data points available in the look-up table will be sliced by **Index** or **Value**.

Index/Value
contains index/values and their ranges.

Index/Value
allows you to specify the discretization index or numerical value of the variable that is being held constant.

Min/Max
are the range of integer values that you are allowed to choose from or display.

Surface Parameters
contains settings where discrete independent variables are held constant and where curve parameters are defined.

Constant Value of
contains the discrete independent variables that can be held constant in the lookup table. The choices are **Scaled Heat Loss/Gain**, **Mean Mixture Fraction**, and **Scaled Variance**. For a two-mixture-fraction case, the **Scaled Heat Loss/Gain** is the only available option. For the FGM model, this group box contains the discrete independent variables that have not been fixed in the **Volume Parameters** group box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Slice by
allows you to select whether the 3D array of data points available in the look-up table will be sliced by **Index** or **Value**.

Index/Value
contains index/values and their ranges.

Index/Value
allows you to specify the discretization index or numerical value of the variable that is being held constant.

Min/Max
are the range of integer values that you are allowed to choose from or display.

Adiabatic
is the enthalpy slice index corresponding to the adiabatic case for which the enthalpy (**Scaled Heat Loss/Gain**) is held constant.

Curve Parameters
allows you to specify the **X-Axis Function** against which the plot variable will be displayed when **2D Curve on 3D Surface** is selected.

X-Axis Function
contains the remaining discrete independent parameters that have not been fixed yet.

Constant Value of *`Independent Parameter`*
allows you to specify the type of discretization for the variable that is being held constant using the controls in the **Slice by** and **Index/Value** group boxes. These items are similar to those described above.

Display
displays the plot variable of the 3D surface.

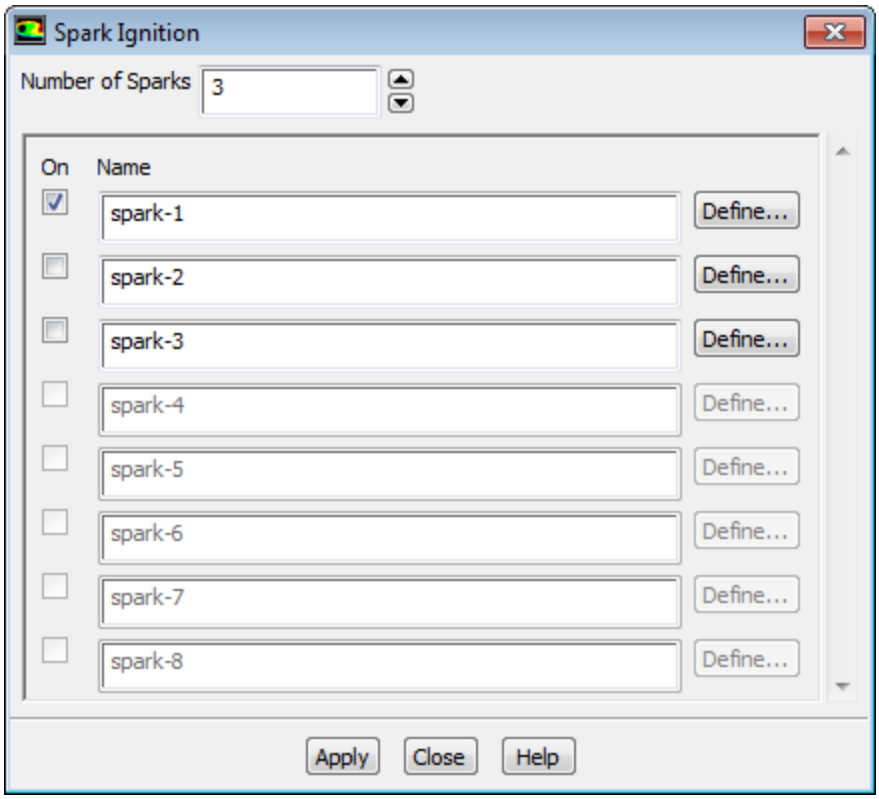
Plot
plots the plot variable for the 2D curve on 3D surface.

Write
opens The Select File Dialog Box (p. 19) where you will specify a name for the file containing plot data. This button appears when **Write To File** is enabled for the **2D Curve on 3D Surface** plot type.

37.4.26. Spark Ignition Dialog Box

The **Spark Ignition** dialog box allows you to define multiple sparks (see Spark Model (p. 1139) for details).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Number of Sparks
is the quantity of sparks you would like to include in your simulation. You can define up to 16 sparks.

On
if enabled, turns on those sparks that will be included in the simulation.

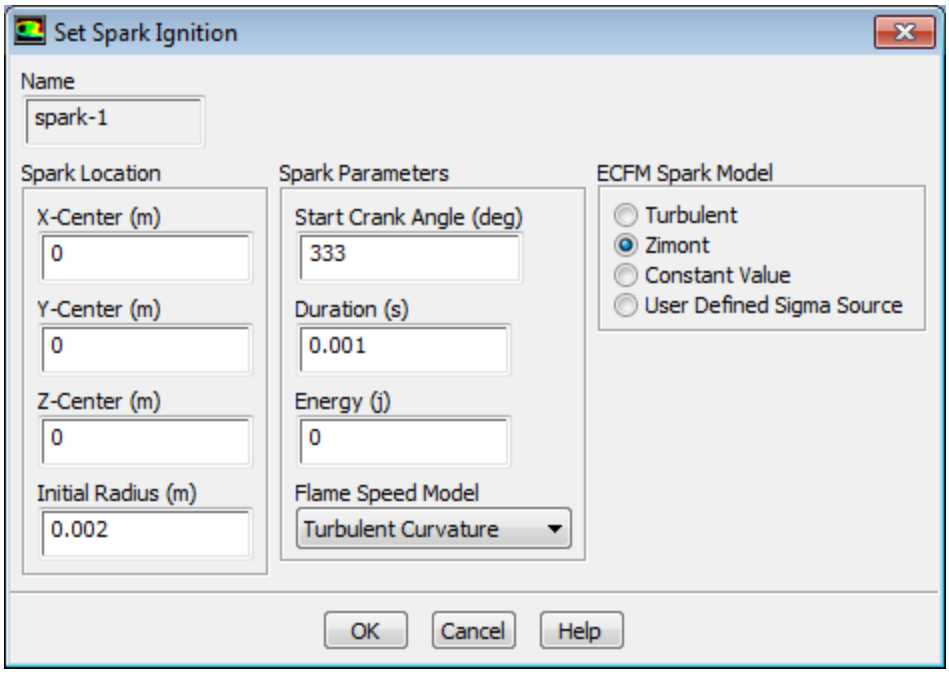
Name
is the name of the spark. You can specify a name, or use the default name.

Define...
opens the Set Spark Ignition Dialog Box (p. 2244).

37.4.27. Set Spark Ignition Dialog Box

The **Set Spark Ignition** dialog box allows you to set the parameters related to the spark ignition model (see Spark Model (p. 1139) for details).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name
displays the name of the spark being defined.

Spark Location
contains the parameters needed to define the location and size of the spark.

X, Y, and Z-Center
specifies x, y, and z coordinates of the spark center.

Initial Radius
specifies the initial spark radius.

Spark Parameters
contains the parameters needed to define the spark.

Start Time
is the time of spark ignition initialization in seconds. When the in-cylinder model is turned on, this control is replaced by the **Start Crank Angle** control.

Start Crank Angle
is the time of spark ignition initialization in crank angle degrees. When the in-cylinder model is turned off, this control is replaced by the **Start Time** option.

Duration
is the duration of the spark ignition in seconds.

Energy
contains the total energy input by the spark. The default energy value is 0. A positive value for energy will cause the temperature of the spark kernel to rise above the combustion process temperature.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Flame Speed Model
allows you to select the turbulent flame speed model for controlling the rate at which the flame front moves.

Turbulent Curvature
includes the effect of flame curvature as specified in Turbulent Curvature in the *Fluent Theory Guide*.

Turbulent Length
neglects the effects of flame curvature on the flame speed as described in Turbulent Length in the *Fluent Theory Guide*.

Herweg-Maly
calculates the turbulent flame speed using Herweg-Maly model. See Herweg-Maly in the *Fluent Theory Guide* for details.

Laminar
specifies the turbulent flame speed as the laminar flame speed. This option can be used to apply user-defined function (UDF) for the turbulent flame speed definition.

Flame Surface Density
allows you to specify the flame surface density as a constant value when **Constant Value** is selected under **ECFM Spark Model**. The value of the flame surface density must be set within the spark region.

User Sigma Source
allows you to apply a custom source term to the ECFM equation within the volume of the spark ignition kernel. For more detail about this user-defined function, refer to Hooking an ECFM Spark Source UDF to ANSYS Fluent in the *Fluent Customization Manual*. This selection is available when **User Defined Sigma Source** is enabled under **ECFM Spark Model**.

ECFM Spark Model
contains model variants used to define the value of the flame surface density. These portions of the dialog box only appear when the **Extended Coherent Flamelet Model** (Setting Up the Extended Coherent Flame Model (p. 1094)) is selected in the **Species Model** dialog box.

Turbulent
takes into account flame wrinkling as described in Turbulent Model in the *Fluent Theory Guide*.

Zimont
operates in the same way as the spark model used with the Zimont combustion model. See Zimont Model in the *Fluent Theory Guide*.

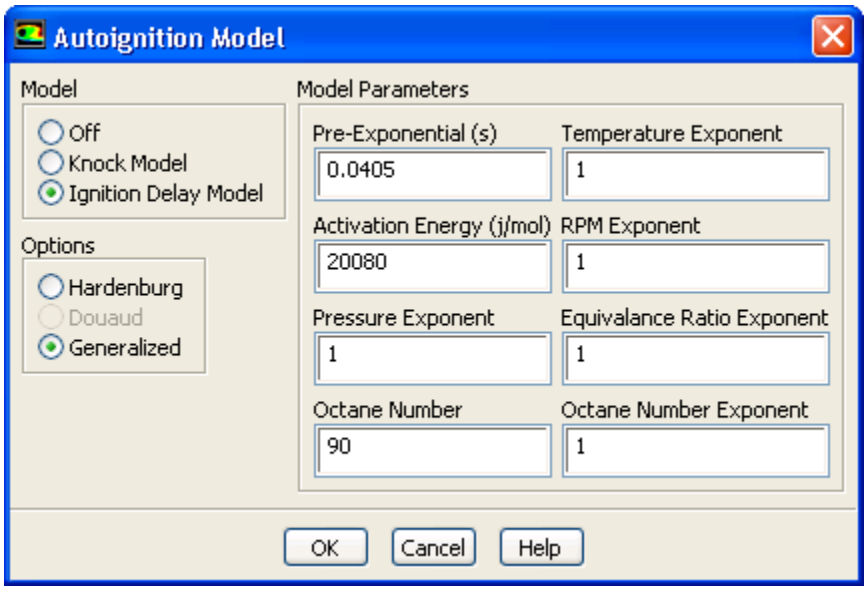
Constant Value
allows you to specify the **Flame Surface Density**.

User Sigma Source
allows you to apply a custom source term to the ECFM equation within the volume of the spark ignition kernel.

37.4.28. Autoignition Model Dialog Box

The **Autoignition Model** dialog box allows you to set the parameters related to the **Knock Model** or the **Ignition Delay Model**. See Modeling Engine Ignition (p. 1139) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Model
contains options to disable or enable models.

Off
disables the model.

Knock Model
enables the knock model. With the **Premixed Combustion** or **Partially Premixed Combustion** models selected, only the **Knock Model** can be turned on.

Ignition Delay Model
enables the ignition delay model.

Options
contains two correlation options that exist with each model.

Douaud
option is used for knock in spark ignition engines. The modeling parameters that are specified for this option are the **Pre Exponential**, **Pressure Exponent**, **Activation Temperature**, **Octane Number**, and **Octane Exponent** (Equation 13.9 in the Theory Guide).

Generalized
enables generalized correlation described by Equation 13.10 in the Theory Guide. It requires the same parameters as in the ignition delay model.

Hardenburg
enables Hardenburg correlation, which is used for heavy-duty diesel engines. This option is enabled only for the ignition delay model.

Model Parameters
contains parameters related to the selected model. See Using the Autoignition Models (p. 1143) for the details about the parameters in this dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pre-Exponential
see Equation 13.10 and Equation 13.9 in the Theory Guide.

Activation Temperature
see Equation 13.9 in the Theory Guide.

Pressure Exponent
see Equation 13.10 and Equation 13.9 in the Theory Guide.

Octane Number
see Equation 13.10 and Equation 13.9 in the Theory Guide.

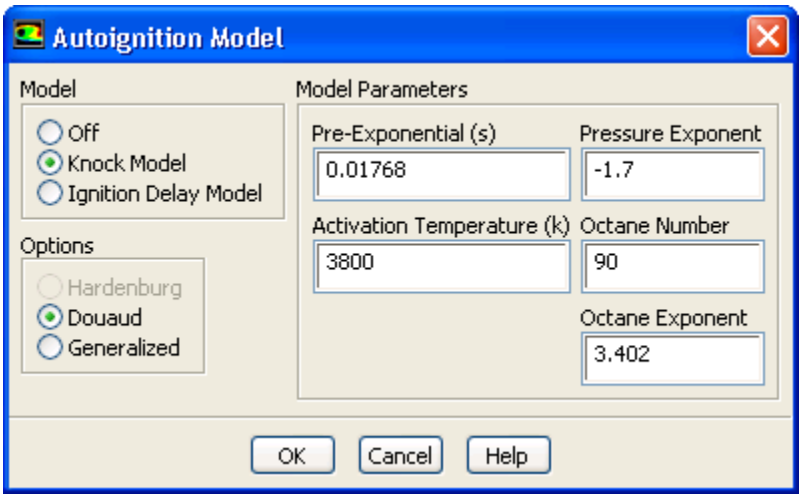
Octane Number Exponent
see Equation 13.10 and Equation 13.9 in the Theory Guide.

Activation Energy
see Equation 13.10 and Equation 13.12 in the Theory Guide.

Temperature Exponent
see Equation 13.10 in the Theory Guide.

RPM Exponent
see Equation 13.10 in the Theory Guide.

Equivalence Ratio Exponent
see Equation 13.10 in the Theory Guide.



Options
contains two correlation options that exist with this model.

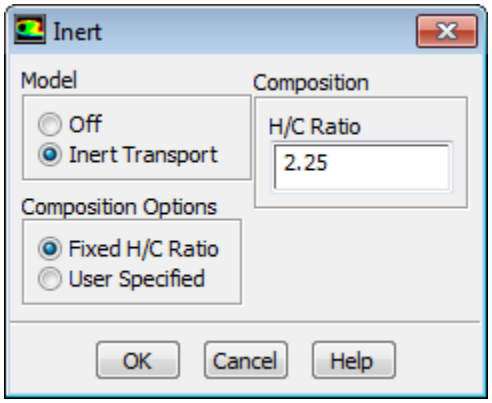
Douaud
enables Douaud correlation (Equation 13.9 in the Theory Guide) used for knock in SI engines.

Generalized
option (Equation 13.10 in the Theory Guide) in the knock model require the same parameters as in the ignition delay model.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.4.29. Inert Dialog Box

The **Inert** dialog box allows you to set the parameters related to the inert model. For details, see Setting Up the Inert Model (p. 1075).



Controls

Model
allows you to enable or disable the inert model.

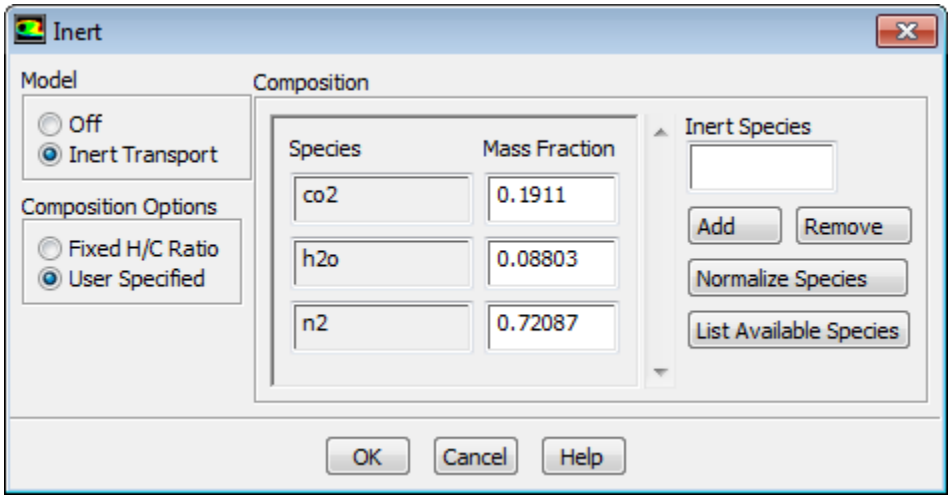
Off
disables the model.

Inert Transport
enables the inert model.

Composition Options
allows you to select a fixed H/C ratio or to specify the composition.

Fixed H/C Ratio
allows you to specify a fixed ratio of hydrogen to carbon in the **H/C Ratio** field.

User Specified
allows you to specify an arbitrary composition for the inert stream.



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Composition
allows you to set the H/C ratio or the mass fraction.

When **Fixed H/C Ratio** is selected under **Composition Options**, the following option(s) are available:

H/C Ratio
specifies the fixed ratio of hydrogen to carbon when the **Fixed H/C Ratio** option is enabled.

When **User Specified** is selected under **Composition Options**, the following option(s) are available:

Species
lists the inert species name.

Mass Fraction
displays the mass fraction of the corresponding species.

Inert Species
allows you to specify the name of the inert species.

Add
adds the specified inert species to the species list.

Remove
removes the specified inert species from the species list.

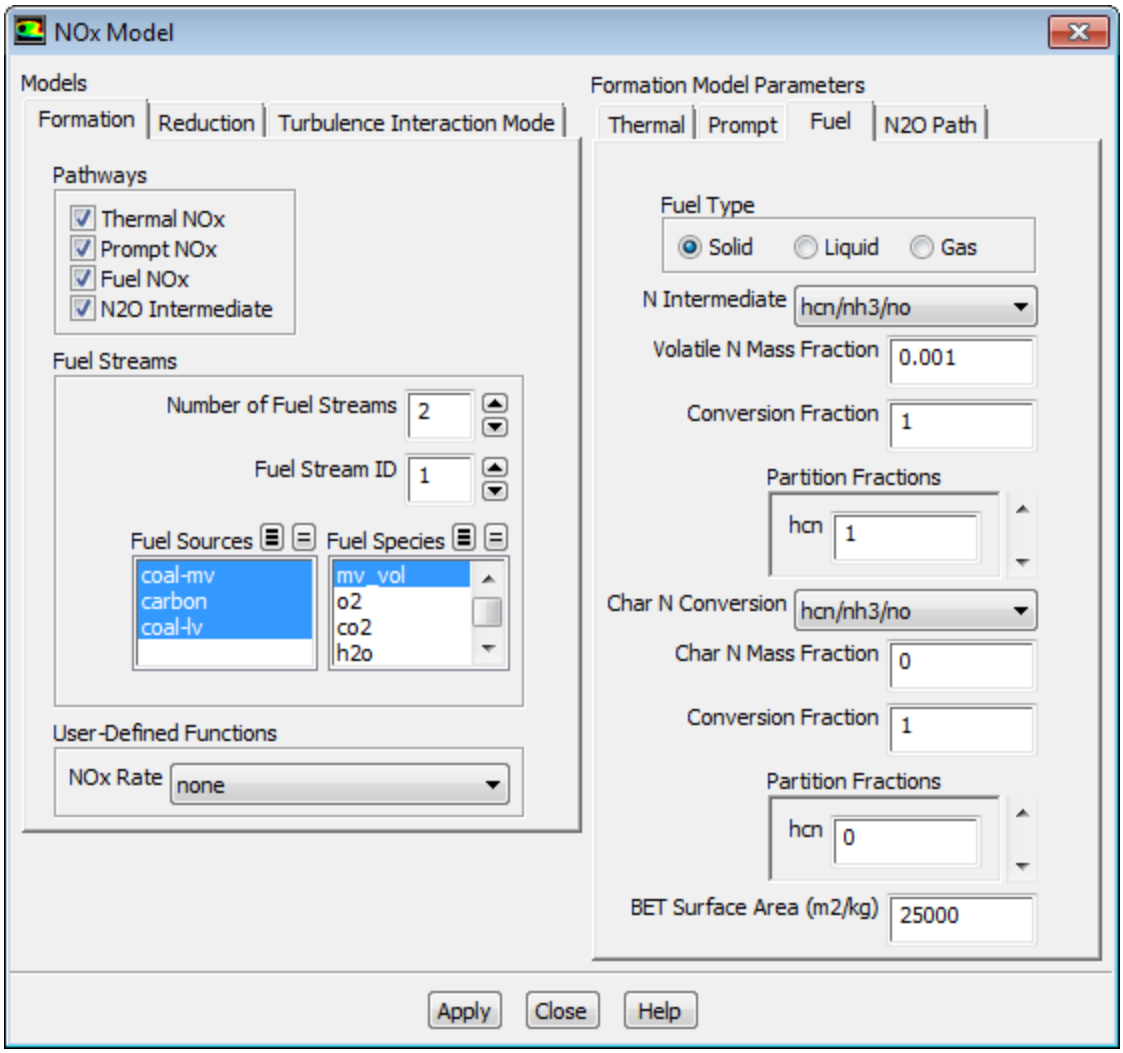
Normalize Species
makes sure the species mass fractions add up to 1.

List Available Species
lists all species in the thermodynamic database file (`thermo.db`) in the console window,

37.4.30. NOx Model Dialog Box

The **NOx Model** dialog box allows you to set parameters related to the NOx postprocessor. See Using the NOx Model (p. 1153) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Models

contains tabs for defining the models used to calculate the NOx production.

Formation

contains the parameters to define the NOx model formation.

Pathways

contains toggle buttons for enabling the NOx models to be used for the calculation of NO and HCN concentrations.

Thermal NOx

enables calculation of thermal NOx.

Prompt NOx

enables the calculation of prompt NOx.

Fuel NOx

enables the calculation of fuel NOx. When using the non-premixed combustion model, the **Fuel NOx** option is only available if the DPM model is also enabled.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

N2O Intermediate
enables the formation of NOx through an N_2O intermediate. This option will only appear if one of the previously listed NOx models is enabled.

Fuel Streams
allows you to define multiple fuel streams for prompt NOx and fuel NOx formation.

Number of Fuel Streams
sets the number of fuel streams. You are allowed up to three fuel streams.

Fuel Stream ID
specifies the fuel stream you are defining in the **PDF Stream** drop-down list, the **Fuel Species** selection list, the **Prompt** tab, and the **Fuel** tab.

PDF Stream
specifies the PDF stream species associated with a particular **Fuel Stream ID**, when calculating fuel NOx formation in conjunction with the non-premixed combustion model. You can select either the **primary** or **secondary** fuel streams, as defined in the PDF table.

Fuel Species
is a list containing all of the defined species, which allows you to specify the species that is the fuel associated with a particular **Fuel Stream ID**. You cannot select more than 5 fuel species for each fuel stream, and the total number of fuel species selected for all the fuel streams combined cannot exceed 10. Note that when the non-premixed combustion model is enabled, your selection in the **Fuel Species** list only applies to prompt NOx calculations.

Fuel Sources
is a list containing all of the fuel N sources. This list is available when **Fuel NOx** is enabled and your case contains injections with different DPM materials defined, such as combusting particles and droplets.

User-Defined Functions
contains the **NOx Rate** drop-down list, which allows you to use a user-defined function (UDF) to contribute to the rate of NOx production. See the separate Fluent Customization Manual for details. Note that you may also use a UDF to specify custom values for the maximum limit (T_{max}) that is used for the integration of the temperature PDF (when temperature is accounted for in the turbulence interaction modeling).

Reduction
allows you to specify the reduction methods.

Methods
contains the list of reduction methods.

Reburn
enables the calculation of NOx reburning effects.

SNCR
enables the calculation of NOx reduction by the SNCR method.

Turbulence Interaction Mode
contains parameters related to the effect of turbulent fluctuations on the NOx formation. See NOx Formation in Turbulent Flows in the Theory Guide for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PDF Mode
is a drop-down list containing options that take into account turbulent fluctuations when you compute the specified NOx formation. See Setting Turbulence Parameters (p. 1166) for details.

none
specifies the use of laminar NOx rate calculations, so that the effects of turbulence are ignored.

temperature
includes fluctuations of temperature.

temperature/species
includes fluctuations of the temperature and the mass fraction of the species selected in the **Species** drop-down list (which appears when you select this option).

mixture fraction
includes fluctuations of the mixture fraction(s). This is available for non-premixed combustion calculations only.

PDF Type
allows you to specify the shape of the PDF.

beta
models the PDF using Equation 14.108 in the Theory Guide.

gaussian
models the PDF using Equation 14.111 in the Theory Guide.

PDF Points
controls the number of points at which the beta function in Equation 14.105 or Equation 14.106 in the Theory Guide will be integrated. The default value of 10, which indicates that the beta function will be integrated at 10 points on a histogram basis. The default value should yield an accurate solution with a reasonable computation time. Increasing this value may improve accuracy, but will also increase the computation time. This field is only available when **temperature** or **temperature/species** is selected from the **PDF Mode** drop-down list.

Temperature Variance
allows you to specify the form of the transport equation that is solved to calculate the temperature variance.

algebraic
is an approximate form of the transport equation (see Equation 14.114 in the Theory Guide).

transported
solves Equation 14.113 in the Theory Guide.

Tmax Option
provides various options for determining the maximum limit(s) for the integration of the PDF used to calculate the temperature.

global-tmax
sets the limit as the maximum temperature in the flow field.

local-tmax-factor
yields cell-based maximum temperature limits by multiplying the local cell mean temperature by the value entered in **Tmax Factor**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

specified-tmax
sets the limit for each cell to be the value entered in **Tmax**.

user-defined
allows you to hook a user-defined function that specifies custom values for the maximum limit (T_{max}), which is used for the integration of the temperature PDF. This option is only available if you have already compiled a user-defined function and selected it in the **Formation** tab.

Species
is a drop-down list that appears when **temperature/species** is selected from the **PDF Mode** drop-down list. Here you will select the species whose mass fraction fluctuations will be factored into the NOx rate calculations.

Formation Model Parameters
contains the tabs used to define the NOx pathways.

Thermal
contains parameters for modeling thermal NOx. (The contents of this tab will appear only if **Thermal NOx** is enabled in the **Formation** tab.)

[O] Model
is a drop-down list in which you can select the method to be used for calculation of thermal NOx. To choose the equilibrium method, select **equilibrium**. To choose the partial equilibrium method, select **partial-equilibrium**. To choose the predicted O concentration method, select **instantaneous**. See Method 1: Equilibrium Approach, Method 2: Partial Equilibrium Approach, and Method 3: Predicted O Approach in the Theory Guide for details.

[OH] Model
is a drop-down list in which you can select the method to be used for calculation of thermal NOx. To exclude OH, select **none**. To choose the partial equilibrium method, select **partial-equilibrium**. To choose the predicted OH concentration method, select **instantaneous**. See Method 1: Exclusion of OH Approach, Method 2: Partial Equilibrium Approach, and Method 3: Predicted OH Approach in the Theory Guide for details.

UDF Rate
provides options for the treatment of the NOx production specified by the user-defined function selected in the **Formation** tab.

Replace Fluent Rate
replaces ANSYS Fluent's thermal NOx rate calculations with the custom NOx rate produced by your user-defined function.

Add to Fluent Rate
adds the custom NOx rate produced by your user-defined function to ANSYS Fluent's thermal NOx rate calculations.

Prompt
contains parameters for modeling prompt NOx. (The contents of this tab will appear only if **Prompt NOx** is enabled in the **Formation** tab.) The settings made in this tab will be associated with a particular fuel stream, specified in the **Fuel Stream ID** field in the **Formation** tab.

Fuel Carbon Number
specifies the number of carbon atoms per fuel molecule.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equivalence Ratio
is the ratio of the actual fuel/air ratio to the stoichiometric fuel/air ratio.

UDF Rate
provides options for the treatment of the NOx production specified by the user-defined function selected in the **Formation** tab.

Replace Fluent Rate
replaces ANSYS Fluent's prompt NOx rate calculations with the custom NOx rate produced by your user-defined function.

Add to Fluent Rate
adds the custom NOx rate produced by your user-defined function to ANSYS Fluent's prompt NOx rate calculations.

Fuel
contains parameters for modeling fuel NOx. (The contents of this tab will appear only if **Fuel NOx** is enabled in the **Formation** tab.) The settings made in this tab will be associated with a particular fuel stream, specified in the **Fuel Stream ID** field in the **Formation** tab.

Fuel Type
specifies the type of fuel NOx to be calculated.

Solid
enables the calculation of solid fuel NOx.

Liquid
enables the calculation of liquid fuel NOx.

Gas
enables the calculation of gas fuel NOx.

N Intermediate
allows you to specify any one of the **hcn**, **nh3**, or **hcn/nh3/no** as the intermediate species. See Fuel NOx Formation in the Theory Guide for details.

Volatile N Mass Fraction
specifies the mass fraction of nitrogen in the volatiles. This parameter appears only for **Solid** fuel NOx calculations.

Fuel N Mass Fraction
specifies the mass fraction of nitrogen in the fuel. This parameter appears only for **Gas** or **Liquid** fuel NOx calculations.

Conversion Fraction
specifies the overall mass fraction of fuel N (for gas and liquid fuels), or volatile N or char N (for solid fuels), that will be converted to intermediate species and/or product NO.

Partition Fractions
specifies the mass fraction of the converted fuel N (for gas and liquid fuels), or volatile N or char N (for solid fuels), that will become **hcn** and **nh3**. The fraction that will become NO will be calculated by the remainder. This option will appear only if you have selected **hcn/nh3/no** for the **N Intermediate** or **Char N Conversion** drop-down lists.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Char N Conversion
is a drop-down list in which you can select **no**, **hcn**, **nh3**, or **hcn/nh3/no** as the species to which the char N is converted (when you are calculating solid fuel NOx). This parameter appears only for **Solid** fuel NOx calculations. See Setting Solid (Coal) Fuel NOx Parameters (p. 1161) for details.

Char N Mass Fraction
specifies the mass fraction of nitrogen in the char. This parameter appears only for **Solid** fuel NOx calculations.

BET Surface Area
sets the BET internal pore surface area (see BET Surface Area in the Theory Guide for details) of the particles. This parameter appears only for **Solid** fuel NOx calculations.

UDF Rate
provides options for the treatment of the NOx production specified by the user-defined function selected in the **Formation** tab.

Replace Fluent Rate
replaces ANSYS Fluent's fuel NOx rate calculations with the custom NOx rate produced by your user-defined function.

Add to Fluent Rate
adds the custom NOx rate produced by your user-defined function to ANSYS Fluent's fuel NOx rate calculations.

N2O Path
contains the method to be used for formation of NO through an N_2O intermediate. (The contents of this tab will appear only if **N20 Intermediate** is enabled in the **Formation** tab.)

N2O Model
contains the drop-down list of available N_2O models.

quasi-steady
enables the quasi-steady-state method of calculation (The transport equation for the species N_2O will not be solved).

transported-simple
enables the transported simple method of calculation (The pollutant species N_2O is added in the species list and it's mass fraction will be calculated using the transport equations).

UDF Rate
provides options for the treatment of the NOx production specified by the user-defined function selected in the **Formation** tab.

Replace ANSYS Fluent Rate
replaces the NOx rate calculated by ANSYS Fluent using N_2O intermediates with the custom NOx rate produced by your user-defined function.

Add to ANSYS Fluent Rate
adds the custom NOx rate produced by your user-defined function to the NOx rate calculated by ANSYS Fluent using N_2O intermediates.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reduction Method Parameters
contains tabs that allow you to define the methods of reduction. (These tabs are do not appear unless a reduction method has been enabled in the **Reduction** tab.)

Reburn
allows you to define the NOx reduction when **Reburn** is enabled in the **Reduction** tab.

Reburn Model
contains the drop-down list of reburn methods.

instantaneous[CH]
enables the instantaneous method in the **Reburn Model**. When you choose this method a warning to include CH, CH_2, and CH_3 will be displayed.

partial-equilibrium
activates partial method in the **Reburn Model**.

Reburn Fuel Species
contains reburn fuel species drop-down list.

Equivalent Fuel Type
contains equivalent fuel type drop-down list.

SNCR
allows you to define the NOx reduction when **SNCR** is enabled in the **Reduction** tab.

Injection Method
contains the parameters for NOx reduction by SNCR method.

gaseous
includes ammonia or urea as a gas-phase pollutant species from the injection locations.

liquid
includes ammonia or urea as a liquid-phase pollutant species from the injection locations.

Reagent Species
allows you to specify the reagent species as either ammonia (**nh3**) or urea (**co<nh2>2**)

Reagent Fraction in Stream
allows you to specify the mass fraction of the reagent in the reagent stream. The remaining mass fraction is assumed to be water. If you enabled a secondary stream in your PDF calculation, by default the secondary stream will act as the reagent stream. Note that the **Reagent Fraction in Stream** field is only available when using the non-premixed combustion model with a liquid-phase reagent injection.

Urea Decomposition
allows you to specify the decomposition model to use when the selected **Reagent Species** is **co<nh2>2**.

rate-limiting
specifies that the source terms be calculated according to the rates given in Table 14.3: Two-Step Urea Breakdown Process.

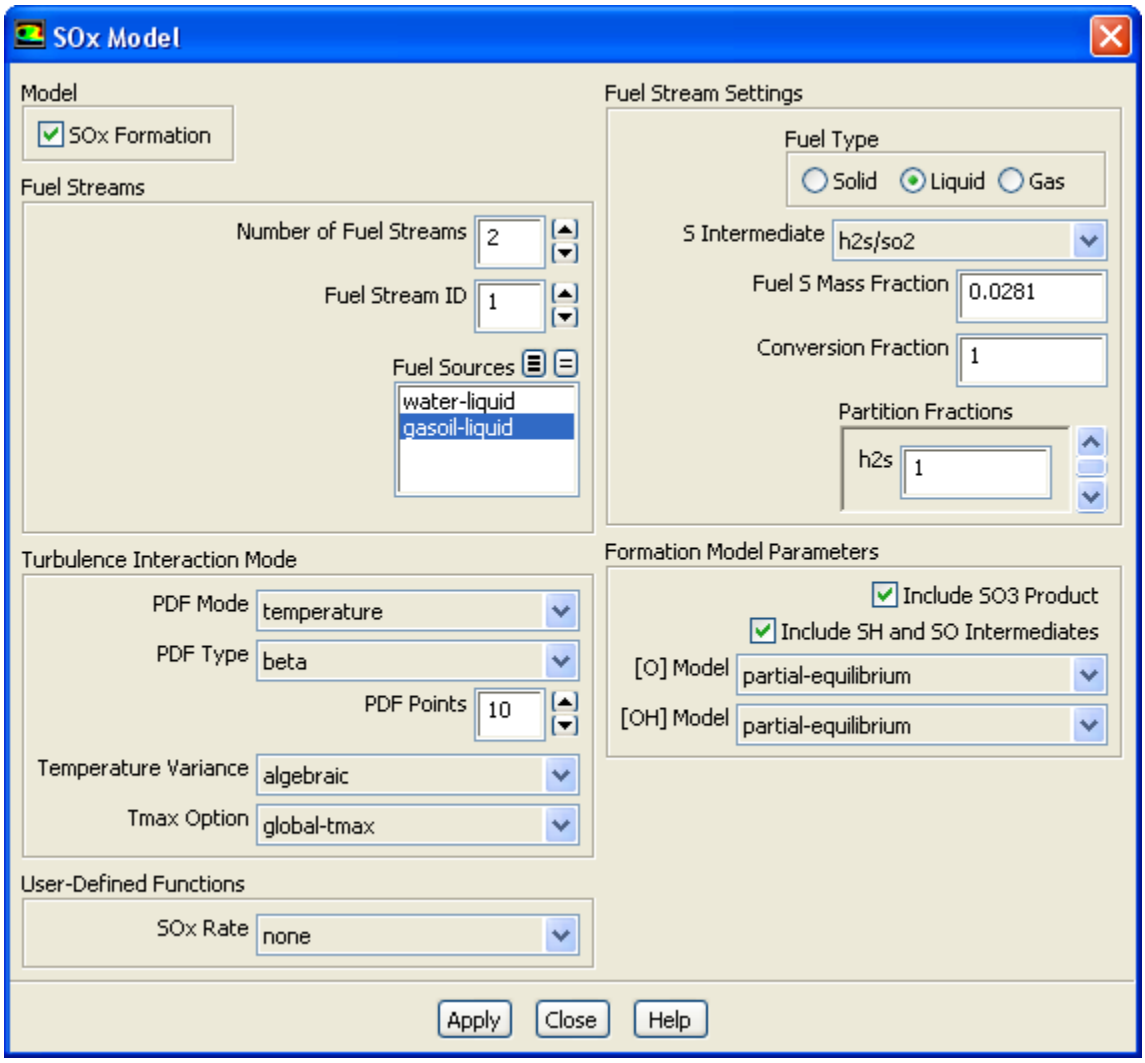
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

user-specified
allows you to specify the molar conversion fraction for ammonia, assuming that the rest of the urea is converted to HNCO.

NH3 Conversion
is the mole fraction of NH_3 in the mixture of NH_3 and HNCO instantly created from the reagent injection. The **NH3 Conversion** field only appears when **user-specified** is selected for **Urea Decomposition**.

37.4.31. SOx Model Dialog Box

The **SOx Model** dialog box allows you to set parameters related to the SOx postprocessor. See Using the SOx Model (p. 1171) for details about the items below.



Controls

Model
contains the control to enable the model.

SOx Formation
enables the model.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fuel Streams
allows you to define multiple fuel streams for SOx formation.

Number of Fuel Streams
sets the number of fuel streams. You are allowed up to three fuel streams.

Fuel Stream ID
specifies the fuel stream you are defining in the **Fuel Species** selection list and the **Formation Model Parameters** group box.

Fuel Species
is a list containing all of the defined species. This list is used to specify the species that is the fuel associated with a particular **Fuel Stream ID**, for any combustion model other than non-premixed combustion. You cannot select more than 5 fuel species for each fuel stream, and the total number of fuel species selected for all the fuel streams combined cannot exceed 10, the list of fuel species is available when **Gas** is the selected **Fuel Type**.

Fuel Sources
is a list containing all of the fuel S sources. This list is available when **Fuel Type** is **Liquid** or **Solid** and your case contains injections with different DPM materials defined, such as combusting particles and droplets.

PDF Stream ID
specifies the PDF stream species associated with a particular **Fuel Stream ID**, when calculating SOx formation in conjunction with the non-premixed combustion model.

primary
indicates the primary fuel stream species, as defined in the PDF table.

secondary
indicates the secondary fuel stream species, as defined in the PDF table.

Turbulence Interaction Mode
contains parameters related to the effect of turbulent fluctuations on the SOx formation. See SOx Formation in Turbulent Flows in the Theory Guide for details.

PDF Mode
is a drop-down list containing options that take into account turbulent fluctuations when you compute the specified SO_2 formation. See Setting Turbulence Parameters (p. 1166) for details.

none
specifies the use of laminar SOx rate calculations, so that the effects of turbulence are ignored.

temperature
includes fluctuations of temperature.

temperature/species
includes fluctuations of temperature and mass fraction of the species selected in the **Species** drop-down list (which appears when you select this option).

mixture fraction
includes fluctuations of the mixture fraction(s). This is available for non-premixed combustion calculations only.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PDF Type
allows you to specify the shape of the PDF.

beta
models the PDF using Equation 14.108 in the Theory Guide.

gaussian
models the PDF using Equation 14.111 in the Theory Guide.

PDF Points
controls the number of points at which the beta function in Equation 14.105 or Equation 14.106 in the Theory Guide will be integrated. The default value of 10, which indicates that the beta function will be integrated at 10 points on a histogram basis, will yield an accurate solution with reasonable computation time. Increasing this value may improve accuracy, but will also increase the computation time. This field is only available when **temperature** or **temperature/species** is selected from the **PDF Mode** drop-down list.

Temperature Variance
allows you to specify the form of transport equation that is solved to calculate the temperature variance.

algebraic
is an approximate form of the transport equation (see Equation 14.114 in the Theory Guide).

transported
solves Equation 14.113 in the Theory Guide.

Tmax Option
provides various options for determining the maximum limit(s) for the integration of the PDF used to calculate the temperature.

global-tmax
sets the limit as the maximum temperature in the flow field.

local-tmax-factor
yields cell-based maximum temperature limits by multiplying the local cell mean temperature by the value entered in **Tmax Factor**.

specified-tmax
sets the limit for each cell to be the value entered in **Tmax**.

user-defined
allows you to hook a user-defined function that specifies custom values for the maximum limit (T_{max}), which is used for the integration of the temperature PDF. This option is only available if you have already compiled a user-defined function and selected it in the **SOx Rate** drop-down list in the **User-Defined Functions** group box.

Species
is a drop-down list which appears when **temperature/species** is selected from the **PDF Mode** drop-down list. Here you will select the species whose mass fraction fluctuations will be factored into the SOx rate calculations.

User-Defined Functions
allows you to use a user-defined function (UDF) to contribute to the rate of SOx production. See the separate Fluent Customization Manual for details. Note that you may also use a UDF to specify custom values

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

for the maximum limit (T_{max}) that is used for the integration of the temperature PDF (when temperature is accounted for in the turbulence interaction modeling).

SOx Rate
allows you to select a compiled user-defined function.

UDF Rate
provides options for the treatment of the SOx production specified by the user-defined function.

Replace Fluent Rate
replaces ANSYS Fluent's SOx rate calculations with the custom SOx rate produced by your user-defined function.

Add to Fluent Rate
adds the custom SOx rate produced by your user-defined function to ANSYS Fluent's SOx rate calculations.

Fuel Stream Settings
contains the parameters associated with a particular fuel stream of the SOx model, as specified in the **Fuel Stream ID** field in the **Fuel Streams** group box.

Fuel Type
enables selection of the fuel.

Solid
enables the calculation of solid fuel SOx.

Liquid
enables the calculation of liquid fuel SOx.

Gas
enables the calculation of gas fuel SOx.

S Intermediate
drop-down list enables you to select intermediate species (**h2s**, **so2**, or **h2s/so2**).

Volatile S Mass Fraction
specifies the mass fraction of sulfur in the volatiles. This parameter appears only for **Solid** fuel streams.

Fuel S Mass Fraction
field sets the value for correct mass fraction of sulfur in the fuel (kg sulfur per kg fuel). This parameter appears only for **Liquid** and **Gas** fuel streams.

Conversion Fraction
specifies the overall mass fraction of the fuel S (for liquid or gas fuels), or the volatile S or char S (for solid fuels), that will be converted to the intermediate species and/or product SO_2. The **S Intermediate Conversion Fraction** has a default value of 1.

Partition Fractions
specifies the mass fraction of the fuel S (for liquid or gas fuels), or the volatile S or char S (for solid fuels) that will become **h2s**. The remainder will become SO_2. This parameter only appears when you select **h2s/so2** for the **S Intermediate** or **Char S Conversion** drop-down lists.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Char S Conversion
drop-down list selects the char S conversion path as **so2**, **h2s**, or **so2/h2s**. This parameter appears only for **Solid** fuel streams.

Char S Mass Fraction
specifies the mass fraction of sulfur in the char. This parameter appears only for **Solid** fuel streams.

Formation Model Parameters
allows you to include SOx products and intermediates.

Include SO3 Product
includes **SO3** as a product in all of the fuel streams, as described in Reaction Mechanisms for Sulfur Oxidation in the Theory Guide.

Include SH and SO Intermediaries
includes **SH** and **SO** as intermediates in all of the fuel streams, as described in Reaction Mechanisms for Sulfur Oxidation in the Theory Guide.

[O] Model
drop-down list specifies the method by which **O** will be calculated in all of the fuel streams, that is, **equilibrium**, **partial-equilibrium**, or **instantaneous** in the **[O] Model**.

[OH] Model
drop-down list specifies the method by which **OH** will be calculated in all of the fuel streams, that is, **equilibrium**, **partial-equilibrium**, or **instantaneous** in the **[OH] Model**.

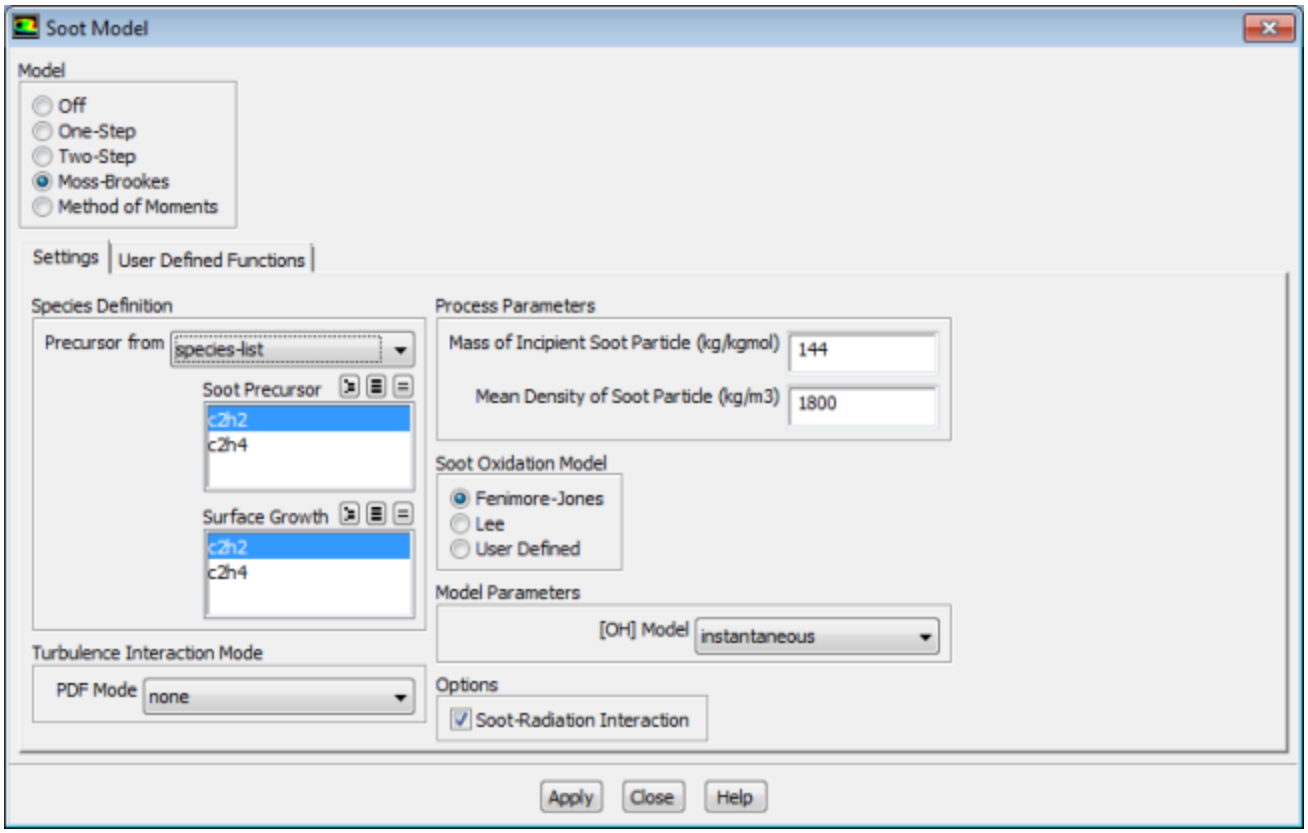
Important

To use the predicted O and/or OH concentration, select **instantaneous** in the **[O] Model** or **[OH] Model** drop-down list.

37.4.32. Soot Model Dialog Box

The **Soot Model** dialog box allows you to set parameters related to the soot model. See Using the Soot Models (p. 1184) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Model

specifies which model should be used for computing soot formation.

Off

disables the calculation of soot formation.

One-Step

enables the one-step soot model described in The One-Step Soot Formation Model in the Theory Guide.

Two-Step

enables the two-step soot model described in The Two-Step Soot Formation Model.

Moss-Brookes

enables the Moss-Brookes soot model described in The Moss-Brookes Model in the Theory Guide.

Moss-Brookes-Hall

enables the Moss-Brookes-Hall soot model described in The Moss-Brookes-Hall Model in the Theory Guide. This option is only available when C_2H_2, C_6H_6, C_6H_5, and H_2 are present in the gas phase species list.

Method of Moment

enables the Method of Moment soot model described in Setting Up the Method of Moments Soot Model (p. 1196).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number of Moments
(only for the method of moments soot model) specifies the number of moments to be used in the Method of Moments soot model. ANSYS Fluent will solve an equal number of moment transport equations. The default value of three moments works reasonably well for wide range of problems. You can specify up to six moments.

Species Definition
contains inputs for specifying the chemical species for your model.

Fuel
is a drop-down list containing all of the defined species. Here you will select the species that is the fuel for the **One-Step** and **Two-Step** models, as well as the **Moss-Brookes** model when a precursor species is not identified in the defined species list.

Oxidant
is a drop-down list containing all of the defined species. Here you will select the species that is the oxidizer for the **One-Step** and **Two-Step** models.

Precursor from
allows you to select from a list of species, enter the correlation values of species, or enables you to hook a user-defined function, used to define the user defined precursor species. This selection is available when using the **Moss-Brookes** and **Moss-Brookes-Hall** models.

User Defined Precursor
allows you to hook a user-defined function to specify the user defined soot precursor. This selection is available when using the **Moss-Brookes** and **Moss-Brookes-Hall** models.

Soot Precursor
is a selection list containing all of the possible precursor species found via a query of the defined species list. By default, ANSYS Fluent only considers c2h2, c6h6, and c2h4 as possible precursor species. For information about including other species in the possible precursor species search, contact your ANSYS Fluent support engineer. From this list you will select the species that are the soot precursor species for the **Moss-Brookes**, **Moss-Brookes-Hall**, or **Method of Moments** models.

Surface Growth
is a selection list containing all of the possible surface growth species, as explained previously for the **Soot Precursor** selection list. Here you will select the species that are the surface growth species for the **Moss-Brookes** and **Moss-Brookes-Hall** models.

Fuel Carbon Number
is the number of carbon atoms in the species selected in the **Fuel** drop-down list. This field appears only for the **Moss-Brookes** and **Moss-Brookes-Hall** model, when **user-correlation** is selected in the **Precursor from** drop-down list.

Fuel Hydrogen Number
is the number of hydrogen atoms in the species selected in the **Fuel** drop-down list. This field appears only for the **Moss-Brookes** and **Moss-Brookes-Hall** model, when **user-correlation** is selected in the **Precursor from** drop-down list.

Molecular Weight of Precursor
is the molecular weight of the precursor species. This field appears only for the **Moss-Brookes** and **Moss-Brookes-Hall** model, when **user-correlation** is selected in the **Precursor from** drop-down list. The default value is the weight of acetylene.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Precursor Correlation
is a drop-down list you can use to define a laminar diffusion profile that relates mixture fraction to precursor mass fraction. This field appears only for the **Moss-Brookes** and **Moss-Brookes-Hall** model, when **user-correlation** is selected in the **Precursor from** drop-down list.

piecewise-polynomial
specifies that the precursor mass fraction is a piecewise-polynomial function of mixture fraction. The default values used by ANSYS Fluent correspond to a methane diffusion flame simulation, in which both the air and fuel initial temperatures are set to 290 K, and acetylene is assumed as the soot precursor. These values can be revised via the **Edit...** button.

constant
specifies that the precursor mass fraction is a constant function of mixture fraction, the value of which is specified in the field below the **Precursor Correlation** drop-down list.

Edit...
opens the Piecewise-Polynomial Profile Dialog Box (p. 2319) when **piecewise-polynomial** is selected from the **Precursor Correlation** drop-down list, thus allowing you to revise the default values.

Sticking Coefficients...
(only for the method of moments soot model) opens Sticking Coefficients Dialog Box (p. 2269) where you can provide the correction factors to be used for calculating the nucleation rates. This button is available only when **User Precursor** is the selected soot nucleation model.

Turbulence Interaction Mode
contains inputs that specify how turbulent fluctuations are accounted for in the soot formation calculations for the **Moss-Brookes** and **Moss-Brookes-Hall** models. For further details on these inputs, see Setting Up the Moss-Brookes Model and the Hall Extension (p. 1190).

PDF Mode
is a drop-down list that contains the options for addressing turbulent fluctuations in the soot rate calculations. Note that **mixture fraction** is the most accurate option, and should be used if it is available.

none
specifies the use of laminar soot rate calculations, so that the effects of turbulence are ignored.

temperature
specifies that the soot rate calculations include the effect of temperature fluctuations.

temperature/species
specifies that the soot rate calculations include the effect of fluctuations of temperature, as well as fluctuations of the mass fraction of the species selected in the **Species** drop-down list (which appears when you select this option).

mixture fraction
is the most accurate option, specifying that the soot rate calculations include the effect of fluctuations of mixture fraction(s). Note that this option is not available if you are using the eddy-dissipation model.

PDF Type
allows you to specify the shape of the PDF.

beta
models the PDF using Equation 14.108 in the Theory Guide.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

gaussian
models the PDF using Equation 14.111 in the Theory Guide.

PDF Points
controls the number of points at which the beta function will be integrated on a histogram basis. Increasing this number may improve accuracy, but will also increase compute time. This field is only available when **temperature** or **temperature/species** is selected from the **PDF Mode** drop-down list.

Temperature Variance
allows you to specify the form of transport equation that is solved to calculate the temperature variance.

algebraic
is an approximate form of the transport equation (see Equation 14.114 in the Theory Guide).

transported
solves Equation 14.113 in the Theory Guide.

Tmax Option
provides various options for determining the maximum limit(s) for the integration of the PDF used to calculate the temperature.

global-tmax
sets the limit as the maximum temperature in the flow field.

local-tmax-factor
yields cell-based maximum temperature limits by multiplying the local cell mean temperature by the value entered in **Tmax Factor**.

specified-tmax
sets the limit for each cell to be the value entered in **Tmax**.

user-defined
allows you to hook a user-defined function that specifies custom values for the maximum limit (T_{max}), which is used for the integration of the temperature PDF. This option is only available if you have already compiled a user-defined function.

Species
is a drop-down list which appears when **temperature/species** is selected from the **PDF Mode** drop-down list. Here you will select the species whose mass fraction fluctuations will be factored into the soot rate calculations.

Process Parameters
contains parameters that control the combustion process modeling.

Mean Diameter of Soot Particle
is the assumed average diameter of the soot particles in the combustion system. For the **Two-Step** model, it is used to compute the soot particle mass m_p in Equation 14.134 in the Theory Guide.

Mean Density of Soot Particle
is the assumed average density of the soot particles in the combustion system. For the **Two-Step** model, it is used to compute the soot particle mass m_p in Equation 14.134 in the Theory Guide. For the **Moss-Brookes** and **Moss-Brookes-Hall** models, it is ρ_{soot} in Equation 14.141 and ρ in Equation 14.144 in the Theory Guide. The default value supplied by ANSYS Fluent is 1800 kg/m^3 (as was used in the work of Brookes and Moss [18] (p. 2887)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Normalization Parameter for Moments
(only for the method of moments soot model) is used to solve the normalized soot moment transport equations to minimize the numerical errors. The default value of this input is sufficient and you may not need to change this.

Stoichiometry for Soot Combustion
is the mass stoichiometry ν_{soot} in Equation 14.131 in the Theory Guide, which computes the soot combustion rate in the **One-Step** and **Two-Step** models. The default value supplied by ANSYS Fluent (2.6667) assumes that the soot is pure carbon and that the oxidizer is O_2.

Stoichiometry for Fuel Combustion
is the mass stoichiometry ν_{fuel} in Equation 14.131 in the Theory Guide, which computes the soot combustion rate in the **One-Step** and **Two-Step** models. The default value supplied by ANSYS Fluent (3.6363) is for combustion of propane ($C_3 H_8$) by oxygen (O_2).

Mass of Incipient Soot Particle
is M_P in Equation 14.142 and Equation 14.144, which is used in the **Moss-Brookes** and **Moss-Brookes-Hall** model computations. The default value supplied by ANSYS Fluent (144 kg/mol) is the mass of 12 carbon atoms. Note that for the original implementation of the Hall extension, the model assumed this mass to be 100 carbon atoms.

Soot Oxidation Model
contains model options that determine the form of the soot oxidation term in the calculations of the **Moss-Brookes** and **Moss-Brookes-Hall** models.

Fenimore-Jones
takes into account the soot oxidation due to the hydroxyl radical.

Lee
takes into account the soot oxidation due to the hydroxyl radical and molecular oxygen.

User Defined
enables you to select a user-defined function for soot oxidation rate. This selection is available when using the **Moss-Brookes** and **Moss-Brookes-Hall** models.

Soot Nucleation Model
(only for the method of moments soot model) contains model options related to soot nucleation modeling.

Use Precursor
enables you to define the precursor species. The nucleation rates are calculated considering coagulation between precursor species using kinetic theory.

Specify Mechanism
enables you to specify nucleation in terms of reactions.

User Defined Oxidation Rate
allows you to hook the user-defined function to specify the soot oxidation rate. This selection is available when using the **Moss-Brookes** and **Moss-Brookes-Hall** models.

Model Parameters
contains parameters that control the soot formation model.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Soot Formation Constant
is the parameter C_s in Equation 14.128 in the Theory Guide. This item appears only for the **One-Step** soot model.

Equivalence Ratio Exponent
is the exponent r in Equation 14.128 in the Theory Guide. This item appears only for the **One-Step** soot model.

Equivalence Ratio Minimum
and **Equivalence Ratio Maximum** are the minimum and maximum values of the fuel equivalence ratio φ in Equation 14.128 in the Theory Guide. Equation 14.128 will be solved only if **Equivalence Ratio Minimum** $< \varphi <$ **Equivalence Ratio Maximum**; if φ is outside of this range, there is no soot formation. This item appears only for the **One-Step** soot model.

Activation Temperature of Soot Formation Rate
is the term E / R in Equation 14.128 in the Theory Guide. This item appears only for the **One-Step** soot model.

Magnussen Constant for Soot Combustion
is the constant A used in the rate expressions governing the soot combustion rate (Equation 14.130 and Equation 14.131) in the Theory Guide. This item appears only for the **One-Step** and the **Two-Step** soot models. For the **Two-Step** model, this input will be called **Magnussen Constant for Soot and Nuclei Combustion**.

Limiting Nuclei Formation Rate
is the limiting value of the kinetic nuclei formation rate η_0 in Equation 14.137 in the Theory Guide. Below this limiting value, the branching and termination term, $(f-g)$ in Equation 14.136 in the Theory Guide, is not included. This item appears only for the **Two-Step** soot model.

Nuclei Branching-Termination Coefficient
is the term $\left(f-g\right)$ in Equation 14.136 in the Theory Guide. This item appears only for the **Two-Step** soot model.

Nuclei Coefficient of Linear Termination on Soot
is the term g_0 in Equation 14.136 in the Theory Guide. This item appears only for the **Two-Step** soot model.

Pre-Exponential Constant of Nuclei Formation
is the pre-exponential term a_0 in the kinetic nuclei formation term, Equation 14.137 in the Theory Guide. This item appears only for the **Two-Step** soot model.

Activation Temperature of Nuclei Formation Rate
is the term E / R in the kinetic nuclei formation term, Equation 14.137 in the Theory Guide. This item appears only for the **Two-Step** soot model.

Alpha for Soot Formation Rate
is α, the constant in the soot formation rate equation, Equation 14.134 **Two-Step** soot model.

Beta for Soot Formation Rate
is β, the constant in the soot formation rate equation, Equation 14.134 in the Theory Guide. This item appears only for the **Two-Step** soot model.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Magnussen Constant for Soot and Nuclei Combustion
is the constant A used in the rate expressions governing the soot combustion rate (Equation 14.130 and Equation 14.131 in the Theory Guide). This item appears only for the **One-Step** and the **Two-Step** soot models.

[OH] Model
is a drop-down list that allows you to specify the method by which the OH radical concentration is calculated, that is, **instantaneous** or **partial-equilibrium**. This list appears only for the **Moss-Brookes**, **Moss-Brookes-Hall**, and **Method of Moments** soot models.

[O] Model
is a drop-down list that specifies the method by which the O radical concentration is calculated, that is, **equilibrium**, **partial-equilibrium**, or **instantaneous**. This list appears only for the **Moss-Brookes** and the **Moss-Brookes-Hall** soot models, when you have selected **partial-equilibrium** for the **[OH] Model**.

Important

To use the concentration of OH or O predicted by the combustion model, select **instantaneous** for **[OH] Model** or **[O] Model**.

Options
contains an option for modeling the effect of soot on a variable radiation absorption coefficient. This group box will appear only when one of the radiation models in the Radiation Model Dialog Box (p. 2190) is active.

Soot-Radiation Interaction
enables the soot-radiation interaction model described in The Effect of Soot on the Absorption Coefficient in the Theory Guide.

Soot Surface Growth Options
(only for the method of moments soot model) contains an option for modeling the soot surface growth.

Site Density
is the number of sites/cm^2 available on the soot surface for surface growth and oxidation reactions with the method of moment soot model.

Enable Aggregation Model
(only for the method of moments soot model) enables the modeling of soot aggregates.

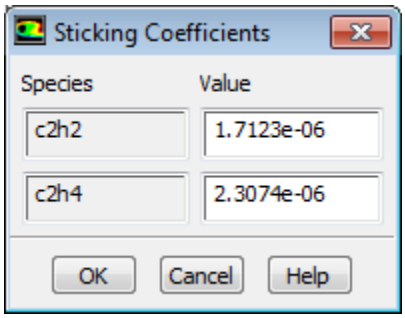
Critical diameter
is the value of the soot particle diameter above which the coagulation regime is switched from coalescent to aggregate.

Fractal dimension
is the parameter that describes the aggregate nature. For soot particles, a value between 1.7 and 2.0 is found to be reasonably good.

37.4.33. Sticking Coefficients Dialog Box

The **Sticking Coefficients** dialog box allows you to set sticking coefficients for the soot model with method of moments. It is opened from the Soot Model Dialog Box (p. 2262). See Setting Up the Method of Moments Soot Model (p. 1196) for details about using this dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

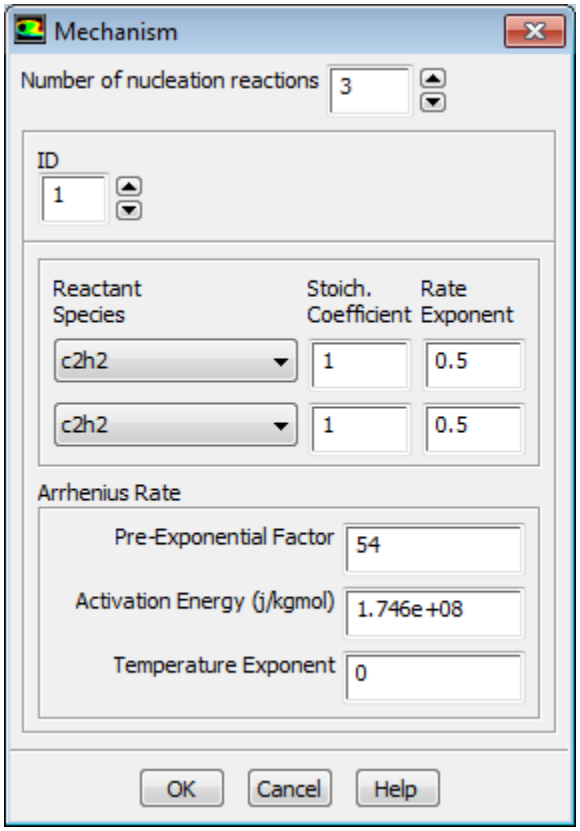


Controls

Species
shows the name of the species for which you specify the **Value**.

37.4.34. Mechanism Dialog Box

The **Mechanism** dialog box allows you to set parameters related to nucleation reactions for the soot model with method of moments. It is opened from the Soot Model Dialog Box (p. 2262). See Setting Up the Method of Moments Soot Model (p. 1196) for details about using this dialog box.



Controls

Number of nucleation reactions
allows you to specify nucleation process as with a single or multi-step reaction mechanism. For each reaction, you must specify all appropriate reaction parameters.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ID
displays the ID of the current nucleation reaction setup.

Reactant Species
is a drop-down list that enables you to select the species for the current nucleation reaction.

Stoich. Coefficient
specifies the stoichiometric coefficient of the reactant species in the nucleation reaction.

Rate Exponent
specifies the rate constant for the reactant species in the current nucleation reaction.

Arrhenius Rate
contains the inputs for the Arrhenius rate parameters related to the current nucleation reaction.

Pre-exponential Factor
is the constant A_r in Equation 7.9 in the Theory Guide. The units of A_r depend on the other rate constant inputs, but must be defined such that the units of the reaction rate $R_{i,r}$ (Equation 7.5 in the Theory Guide) are in (kg/m^3-s) if you are using SI units.

Important

It is important to note that if you have selected the British units system, the Arrhenius factor should still be specified in SI units. This is because ANSYS Fluent applies no conversion factor to the value you enter for A_r (the conversion factor is 1.0) when you work in British units, as the correct conversion factor depends on your values for $\nu_{j,r}$, β_r, and so on.

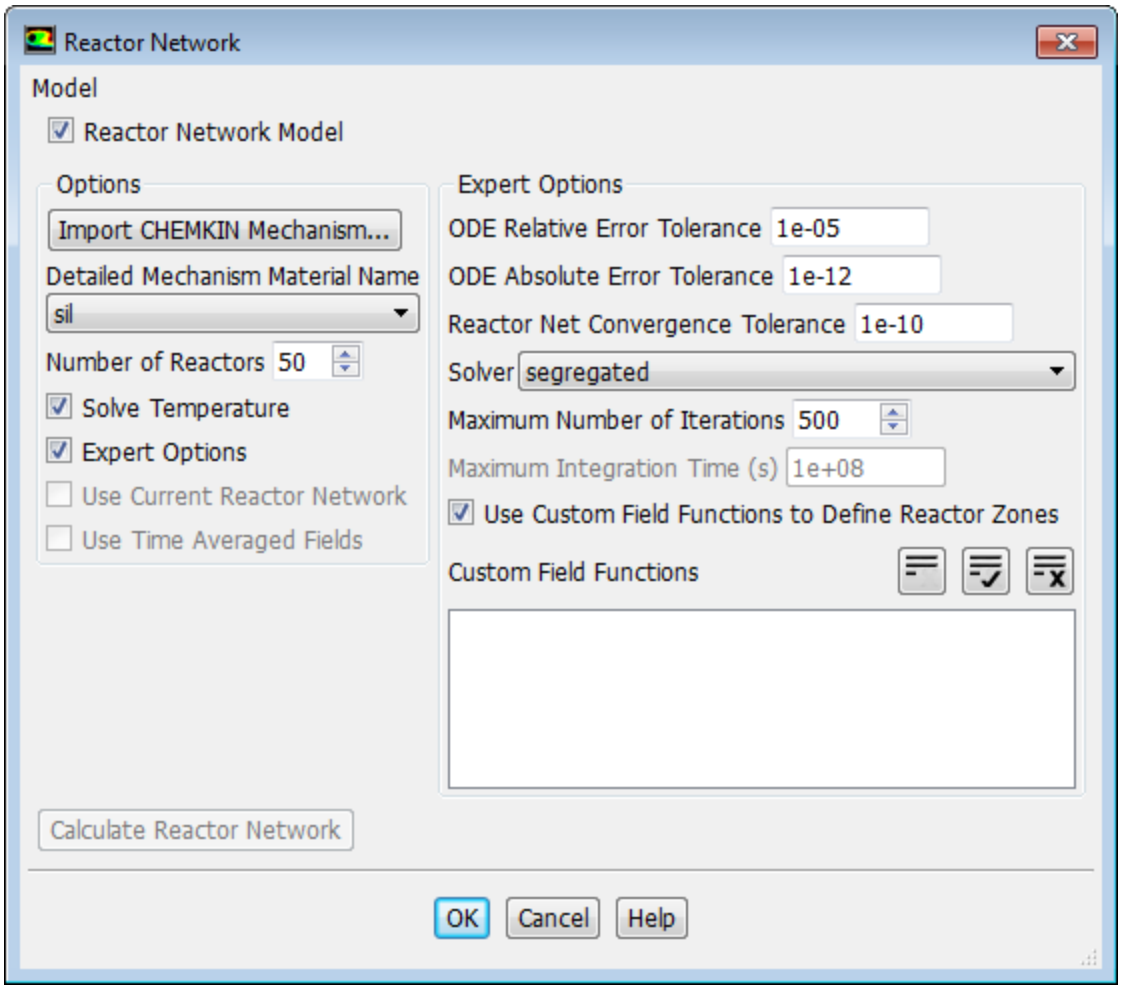
Activation Energy
is the constant E_r in the forward rate constant expression, Equation 7.9 in the Theory Guide).

Temperature Exponent
is the value for the constant β_r in Equation 7.9 in the Theory Guide.

37.4.35. Reactor Network Dialog Box

The **Reactor Network** dialog box allows you to set parameters related to the reactor network model and calculate the reactor network for species transport/combustion. See Reactor Network Model (p. 1020) for details about using this dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Model

contains the option for enabling (or disabling) the reactor network model.

Reactor Network Model

enables/disables the modeling of the reactor network.

Options

contains the options for reactor network model. This portion of the dialog box appears only if **Reactor Network Model** is selected.

Import CHEMKIN Mechanism

allows you to import a detailed chemical mechanism in CHEMKIN format.

Detailed Mechanism Material Name

is a drop-down list of available mixture materials from which you can select a mixture material for your reactor network simulation.

Number of Reactors

sets the number of reactors in which cells are grouped based on specified criteria.

Solve Temperature

specifies whether or not the reactor network algorithm solves the temperature. By default, this option is selected.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Expert Option
when selected, displays the advanced options related to the solver settings.

Use Current Reactor Network
allows you to perform additional reactor network simulations with the existing network of connected reactors. This option can be enabled once the Fluent solver agglomerates cells and obtains the reactor-network solution.

Use Time Averaged Fields
enables you to select the time-averaged composition fields to be used for clustering reactors and the time-averaged velocity fields for calculating reactor mass flux matrix. This option is available only for unsteady simulations with enabled **Data Sampling For Unsteady Statistics** option.

Expert Options
contains advanced options for ODE solver and for reactor cell clustering control. This portion of the dialog box appears only if **Expert Options** is selected.

ODE Relative Error Tolerance
specifies relative error control for ODE integrator.

ODE Absolute Error Tolerance
specifies absolute error control for ODE integrator.

Reactor Network Convergence Tolerance
allows you to specify tolerance to control the accuracy of the reactor network calculations.

Solver
allows you to select the solver for your simulation. The default segregated solver typically converges faster than the coupled solver. However, if residuals stall above an acceptable tolerance, the coupled solver should be used.

Maximum Number of Iterations
(segregated solver only) sets the maximum number of solver iterations.

Maximum Integration Time
(coupled solver only) sets the maximum elapsed run time for ODE solver. ANSYS Fluent terminates the run at this time if the residuals fail to converge.

Use Custom Field Functions to Define Reactor Zones
enables the use of custom field functions (Custom Field Functions (p. 2003)) for reactor network cell clustering.

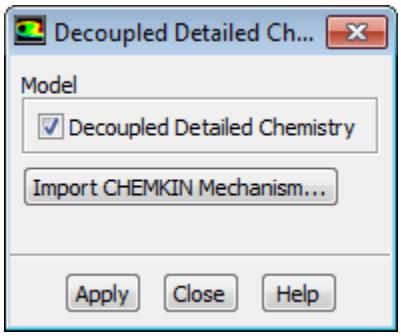
Custom Field Functions
contains the list of available user-defined field functions from which you can select the functions for reactor network cell clustering. This selection list will not appear if the **Use Custom Field Functions to Define Reactor Zones** option is not selected.

Calculate Reactor Network
calculates a chemically reactive flow using reactor network model.

37.4.36. Decoupled Detailed Chemistry Dialog Box

The **Decoupled Detailed Chemistry** dialog box allows you to postprocess slowly-forming, trace pollutant species on a steady-state flow field using detailed chemical kinetic mechanisms.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Model
includes the option to enable **Decoupled Detailed Chemistry**.

Import CHEMKIN Mechanism...
opens the CHEMKIN Mechanism Import Dialog Box (p. 2642).

Integration Parameters...
opens the Integration Parameters Dialog Box (p. 2235).

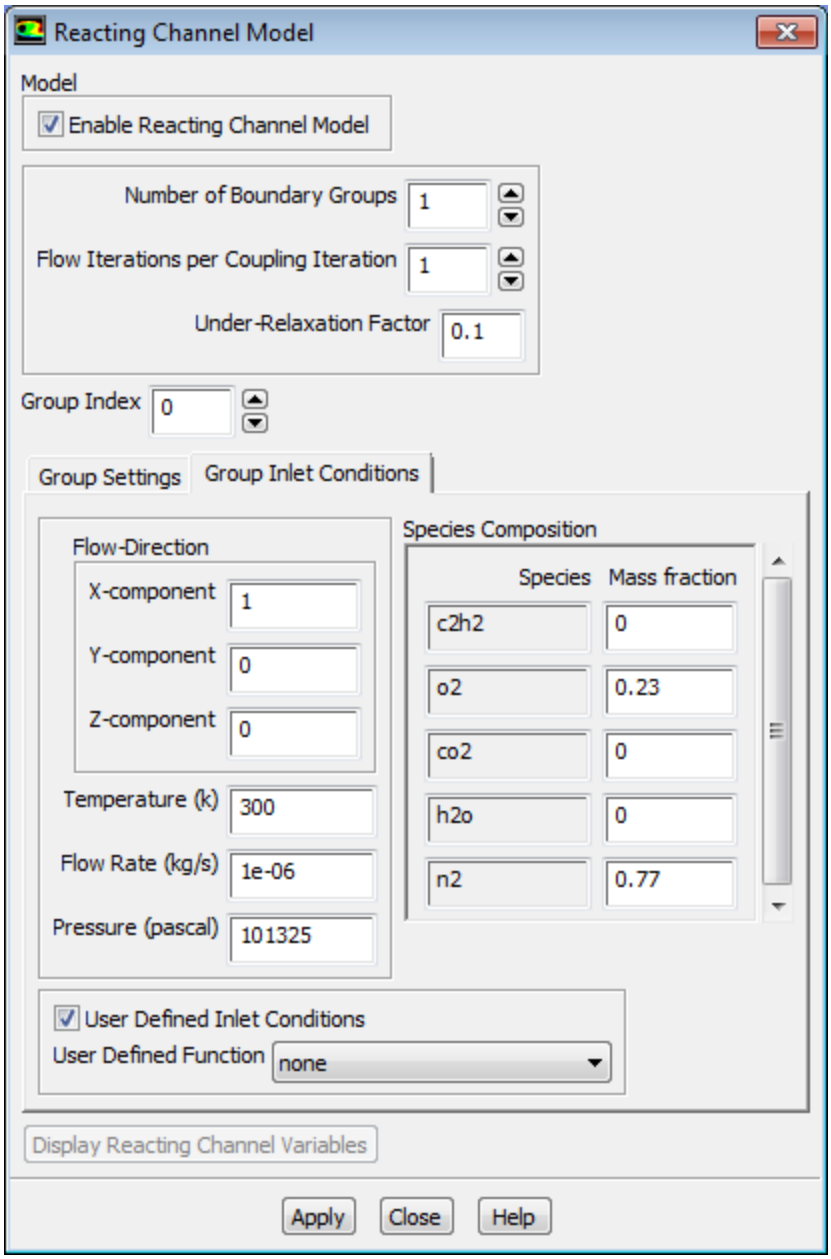
Original Species
are the species that are in the original case setup.

Pollutant Species
are typically slowly forming (far from chemical equilibrium), and occur at miniscule mass fractions.

37.4.37. Reacting Channel Model Dialog Box

The **Reacting Channel Model** dialog box allows you to solve reacting flow in shell and tube heat exchangers with long and thin channels.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Model

includes the option to **Enable Reacting Channel Model**.

Number of Boundary Groups

allows you to group together channels with common flow direction, mixture materials, inlet compositions, temperature, pressure, and mass flow rate in cases where you have multiple channels.

Flow Iterations per Coupling Iteration

is the number of outer flow iterations for each channel flow iteration.

Under-Relaxation Factor

is used to update the heat flux from the reacting channel. See Equation 7.107 in the Theory Guide.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Group Index
is used to identify the boundary group.

Group Settings
contains **Group Settings** that you will set.

Channel Walls in Group
are wall boundary zones that correspond to the reacting channels in the group.

Material
is the group material.

Import CHEMKIN Mechanism...
opens the CHEMKIN Mechanism Import Dialog Box (p. 2642).

Model Options
allows you to enable **Surface Reactions** and **Porous Medium** models and set related parameters.

Group Inlet Conditions
is where you specify the **Temperature**, the **Flow Rate**, the **Pressure**, and other inlet conditions of the reacting channel group.

Flow Direction
is the **X-**, **Y-**, and **Z-component** of the flow at the inlets of the channels in the current group.

Species Composition
is where you will specify the inlet mass fractions of the group.

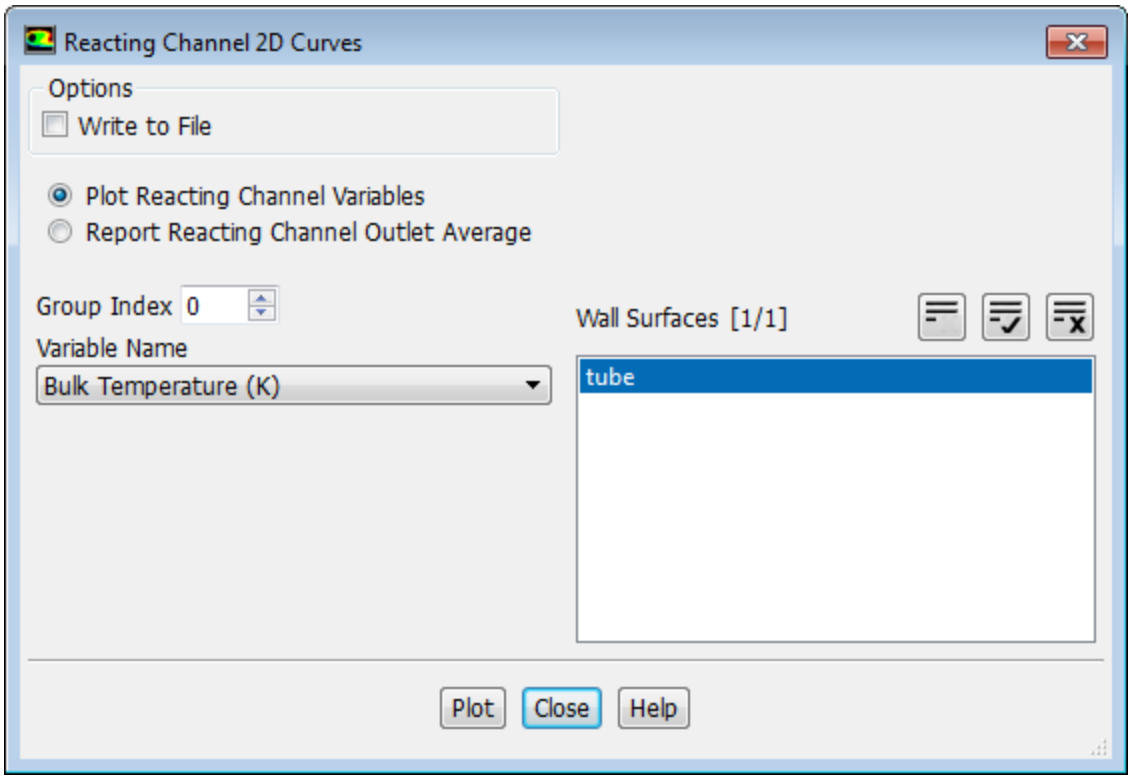
User Defined Inlet Conditions
enables you to hook a user-defined function to specify the reacting channel inlet conditions. When this option is selected, the **User Defined Function** drop-down list becomes visible.

Display Reacting Channel Variables
opens the Reacting Channel 2D Curves Dialog Box (p. 2276) for postprocessing reacting channel variables.

37.4.38. Reacting Channel 2D Curves Dialog Box

The **Reacting Channel 2D Curves** dialog box allows you to display or write 2D curves of the reacting channel.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options
gives you the option to **Plot**, **Report**, or **Write to File** channel variables.

Group Index
is used to identify the reacting channel boundary group.

Plot Reacting Channel Variables
enables the plotting of reacting channel variables.

Report Reacting Channel Outlet Average
enables the reporting of reacting channel outlet average.

Variable Name
specifies the channel variable for plotting.

Variable Names
contains a selectable list of channel variables for reporting.

Wall Surfaces
contains a selectable list of wall surfaces on which you can plot, report or write the selected channel variables.

Plot
generates an X-Y plot. The **Plot** button is available when **Plot Reacting Channel Variables** is selected. When the **Write to File** option is enabled, the **Plot** button changes to the **Write...** button.

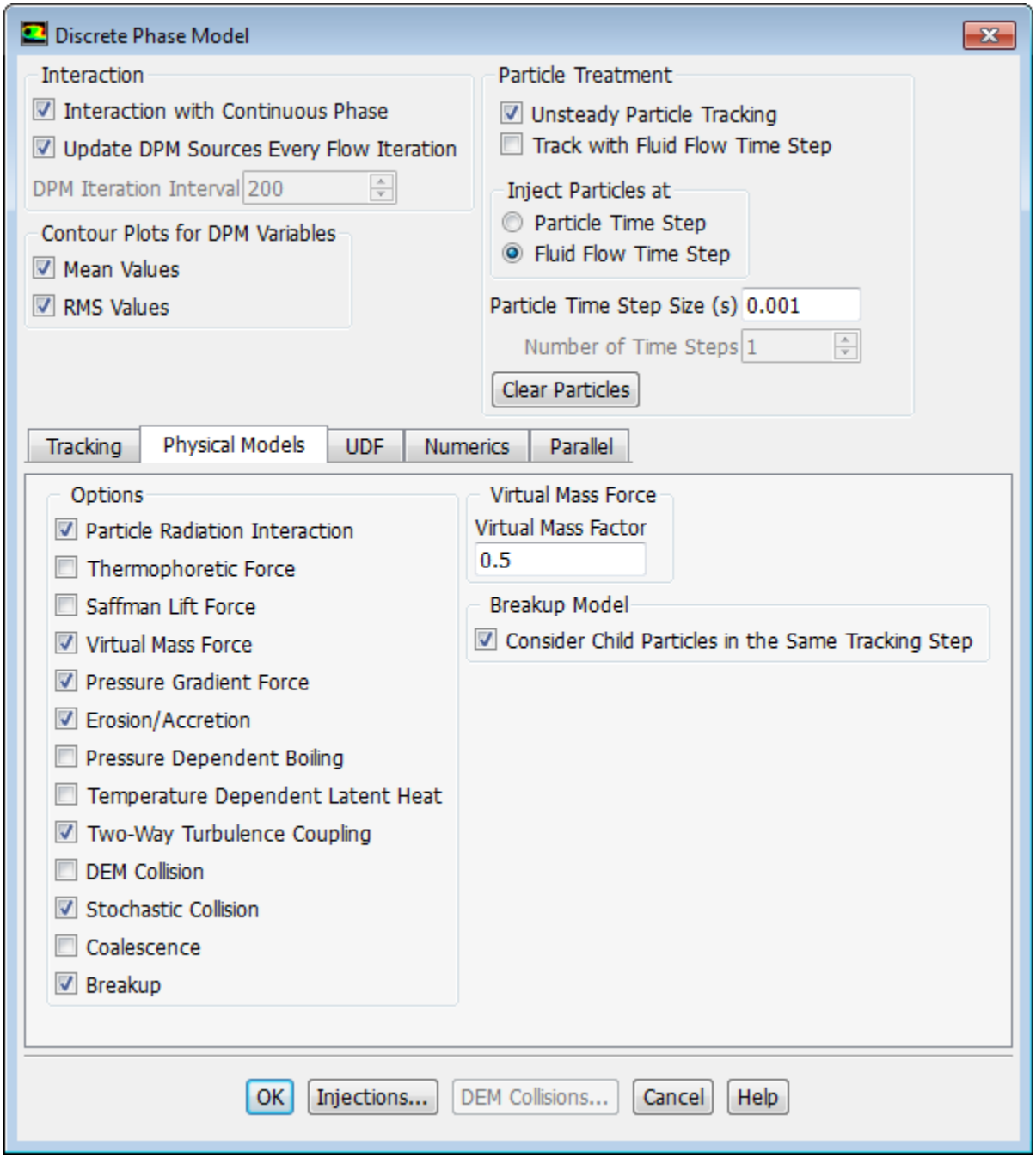
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Report

generates the report of the channel variables. The **Report** button is available when **Report Reacting Channel Outlet Average** is selected. When the **Write to File** option is enabled, the **Report** button changes to the **Write...** button.

37.4.39. Discrete Phase Model Dialog Box

The **Discrete Phase Model** dialog box allows you to set parameters related to the calculation of a discrete phase of particles. See Modeling Discrete Phase (p. 1235) for details.



Controls

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Interaction
contains parameters used for performing coupled calculations of the continuous and discrete phase flow. See Procedures for a Coupled Two-Phase Flow (p. 1327) for details.

Interaction with Continuous Phase
enables a coupled calculation of the discrete phase and the continuous phase.

Update DPM Sources Every Flow Iteration
enables calculation of particle source terms at every DPM Iteration. For unsteady simulations, it is the default and is recommended.

DPM Iteration Interval
allows you to control the frequency at which the particles are tracked and the DPM sources are updated.

Contour Plots for DPM Variables
contains options for enabling cell-averaged discrete phase variables for postprocessing.

Mean Values
enables additional cell-averaged discrete phase variables for postprocessing.

RMS Values
enables additional RMS values for several discrete phase variables for postprocessing.

Particle Treatment
contains options for choosing to treat the particles in an unsteady or a steady fashion.

Unsteady Particle Tracking
enables unsteady tracking of particles.

Track with Fluid Flow Time Step
enables the use of fluid flow time steps to inject the particles.

Inject Particles at
contains parameters to decide when to inject the particles for a new time step.

Particle Time Step
enables injection of particles for every particle time step.

Fluid Flow Time Step
enables injection of particles for every fluid flow time step.

In any case, the particles will always be tracked in such a way that they coincide with the flow time of the continuous flow solver, as long as the **Max. Number of Steps** (in the **Tracking** tab) used to compute a single trajectory is sufficient.

Particle Time Step Size
specifies particle time step size for the calculation.

Number of Time Steps
allows you to specify the number of time steps for the calculation.

Clear Particles
clears the particles that are currently in the domain.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tracking
contains two parameters to control the time integration of the particle trajectory equations.

Tracking Parameters
contains parameters that control the tracking of particle trajectories. For steady-state particle tracking, one simple rule of thumb to follow when setting the two parameters below is that if you want the particles to advance through a domain of length D, the **Length Scale** times the number of **Max. Number of Steps** should be approximately equal to D. See Integration of Particle Equation of Motion in the Theory Guide for details about the items below.

Max. Number of Steps
is the maximum number of time steps used to compute a single particle trajectory via integration of Equation 16.1 in the Theory Guide and Equation 16.58. The default value is 50,000 for steady-state particle tracking and 500 for unsteady particle tracking.

Specify Length Scale
when enabled allows you to specify the length scale.

Length Scale
controls the integration time step size used to integrate the equations of motion for the particle. It appears only when the **Specify Length Scale** option is enabled.

Step Length Factor
specifies the value of λ in Equation 23.3 (p. 1248).

Physical Models
contains optional discrete phase models and their relevant parameters.

Options
contains additional models that can be included in the calculation. See Physical Models for the Discrete Phase Model (p. 1249) for more information.

Particle Radiation Interaction
includes the effect of radiation heat transfer to the particles (Equation 5.34 in the Theory Guide). You will also need to define additional properties for the particle materials (emissivity and scattering factor), as described in Description of the Properties (p. 1317).

This item appears only if the P-1 or discrete ordinates model is selected in the Radiation Model Dialog Box (p. 2190).

Thermophoretic Force
enables the inclusion of a thermophoretic force on the particles as an additional force term. See Thermophoretic Force in the Theory Guide for details.

Saffman Lift Force
enables the inclusion of Saffman's lift force (lift due to shear) as an additional force term. See Saffman's Lift Force in the Theory Guide for details.

Virtual Mass Force

enables the inclusion of virtual mass forces in the particle force balance. See Including the Virtual Mass Force and Pressure Gradient Effects on Particles (p. 1251) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pressure Gradient Force

enables the inclusion of pressure gradient effects in the particle force balance. See Including the Virtual Mass Force and Pressure Gradient Effects on Particles (p. 1251) for details.

Erosion/Accretion

enables the monitoring of erosion/accretion rates at wall boundaries. See Monitoring Erosion/Accretion of Particles at Walls (p. 1251) for details. This item appears only if **Interaction with Continuous Phase** is enabled.

Pressure Dependent Boiling

allows you to switch from droplet vaporization (Law 2) to boiling (Law 3), as described in Enabling Pressure Dependent Boiling (p. 1251).

Temperature Dependent Latent Heat

allows you to include the droplet temperature effects on the latent heat, as described in Equation 16.111 in the Theory Guide.

Two-Way Turbulence Coupling

enables the effect of change in turbulent quantities due to particle damping and turbulence eddies.

DEM Collision

allows you to model DEM collision as described in Modeling Collision Using the DEM Model (p. 1253).

Stochastic Collision

enables the effect of droplet collisions, as described in Collision and Droplet Coalescence Model Theory in the Theory Guide.

Coalescence

enables the effect of droplet coalescence, as described in Collision and Droplet Coalescence Model Theory in the Theory Guide. This option is available when the **Stochastic Collision** option is enabled.

Breakup

enables breakup for all suitable injections. You can select the breakup model and parameters, or disable breakup, for each injection in the Set Injection Properties Dialog Box (p. 2796).

Breakup Model

contains an option to **Consider Child Particles in the Same Tracking Step**. This control appears only when **Breakup** is enabled under **Options**.

DEM Collision Model

contains model settings that allow you to continue your setup of the model.

Adaptive Collision Mesh Width

is enabled by default. This adjusts the width of the collision mesh to the largest parcel diameter multiplied by the **Edge Scale Factor**

Edge Scale Factor

is the factor by which the width of the collision mesh is adjusted to the largest parcel diameter.

Static Collision Mesh Width

is the fixed width of the collision mesh. This appears when the **Adaptive Collision Mesh Width** is disabled.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Maximum Particle Velocity
limits the maximum particle velocity to a physically plausible range.

DEM Collisions...
opens the DEM Collisions Dialog Box (p. 2285). This button is available at the bottom of the **Discrete Phase Model** dialog box when the **DEM Collision** model is enabled.

UDF
contains parameters that can be used to customize the discrete phase model using a user-defined function. See the separate Fluent Customization Manual for details about user-defined functions.

User-Defined Functions
lists will show available user-defined functions that can be selected to customize the discrete phase model.

Body Force
contains a drop-down list of user-defined functions available for including additional body forces.

Erosion/Accretion
contains a drop-down list of user-defined functions available for incorporating non-standard erosion rate definitions. This item will appear only when the **Erosion/Accretion** option has been enabled.

Scalar Update
contains a drop-down list of user-defined functions available for calculating or integrating scalar values along the particle trajectory.

Source
contains a drop-down list of user-defined functions available for modifying interphase exchange terms.

Spray Collide Function
contains a drop-down list of user-defined functions available for modifying spray collide function.

DPM Timestep
contains a drop-down list of user-defined functions available for modifying DPM time step.

Impingement Model
contains a drop-down list of user-defined functions available for modifying the impingement model. If selected, the **Impingement Model** UDF will overwrite the impingement/splashing model selected for the Lagrangian or Eulerian wall film boundary condition. This item is available only when either **wall-film** is selected for **Boundary Cond. Type** in the **DPM** tab of the **Wall** dialog box or the **Eulerian Wall Film** model is enabled.

Film Regime
contains a drop-down list of user-defined functions available for setting the film regime of particle impingement. This item is available only when either **wall-film** is selected for **Boundary Cond. Type** in the **DPM** tab of the **Wall** dialog box or the **Eulerian Wall Film** model is enabled.

Splashing Distribution
contains a drop-down list of user-defined functions available for modifying distribution of splashed particle variables. If selected, the **Splashing Distribution** UDF will overwrite the impingement/splashing model selected for the Lagrangian or Eulerian wall film boundary condition. This item is available only when either **wall-film** is selected for **Boundary Cond. Type** in the **DPM** tab of the **Wall** dialog box or the **Eulerian Wall Film** model is enabled.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

User Variables
lists the user input variables.

Number of Scalars
sets the number of scalar values used in the calculations with user-defined functions.

Numerics
tab gives you control over the numerical schemes for particle tracking as well as solutions of heat and mass equations. See Numerics of the Discrete Phase Model (p. 1260) for details.

Tracking Options
contains parameters of solutions of heat and mass equations.

Accuracy Control
enables the solution of equations of motion within a specified tolerance.

Tolerance
is the maximum relative error that has to be achieved by the tracking procedure.

Max. Refinements
is the maximum number of step size refinements in one single integration step. If this number is exceeded the integration will be conducted with the last refined integration step size.

Track in Absolute Frame
enables tracking the particles in the absolute reference frame.

Tracking Scheme Selection
contains parameters for selection of numerical schemes.

Automated
enables a mechanism to switch in an automated fashion between numerically stable lower order schemes and higher order schemes, which are stable only in a limited range.

High Order Scheme
can be chosen from the group consisting of **trapezoidal** and **runge-kutta** scheme.

Low Order Scheme
consists of **implicit** and the exponential **analytic** integration scheme.

Tracking Scheme
allows you to choose any of the tracking schemes. You also can combine each of the tracking schemes with Accuracy Control. It is selectable only if **Automated** is switched off.

Coupled Heat-Mass Solution
enables the solution of the corresponding equations using a coupled ODE solver with error tolerance control for the **Droplet**, **Combusting**, or **Multicomponent** particles. See Including Coupled Heat-Mass Solution Effects on the Particles (p. 1262) for details.

Vaporizing Limiting Factors
for the **Mass** and **Heat** are set at the default values of 0.3 and 0.1, respectively. See Including Coupled Heat-Mass Solution Effects on the Particles (p. 1262) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Averaging

contains settings for node based averaging of DPM quantities. See Node Based Averaging of Particle Data (p. 1263) for details.

Enable Node Based Averaging
enables node based averaging and displays additional related options.

Average DPM Source Terms
enables averaging of source terms for the discrete phase.

Average in Each Integration Time Step
enables averages during each integration time step.

Average DDPM Variables
enables averaging of DDPM variables when the DDPM model is active.

Kernel Settings

contains settings for the kernel used by the node based averaging algorithm.

Averaging Kernel
selects the kernel to use for averaging

Gaussian Factor
sets the constant for the Gaussian distribution when **gaussian** is selected for **Averaging Kernel**.

Source Terms

Linearize Source Terms
enables the linearization of source terms for the discrete phase.

Parallel
contains parameters that control the compute nodes for performing discrete phase calculations in parallel. See Parallel Processing for the Discrete Phase Model (p. 1360) for details.

Methods
allows you to select the mode for parallel processing. This group box is editable only for the parallel version of ANSYS Fluent; for the serial version, the **Shared Memory** option (see the description that follows) is selected by default.

Message Passing
enables cluster computing and also works on shared-memory machines. With this option, the compute node processes themselves perform the particle work on their local partitions and particle migration to other compute nodes is implemented using message passing primitives. No special requirements are placed on the host machine. Note that this model is not available if the **Cloud Model** option is enabled in the **Turbulent Dispersion** tab of the **Set Injection Properties** dialog box. The **Message Passing** option is irrelevant for the serial ANSYS Fluent code while the **Shared Memory** option remains valid.

Shared Memory
allows you to specify parameters for performing the calculations on shared-memory multiprocessor machines.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Hybrid
allows you to combine **Message Passing** (as previously described) and **OpenMP** for a dynamic load balancing without migration of cells. This option enables multicore cluster computing, and also works on shared-memory machines. When using the **Hybrid** option, you can control the maximum number of threads on each machine via the **Thread Control** dialog box (see Controlling the Threads (p. 2058) for details). Note that this option is not available if the **Cloud Model** option is enabled in the **Turbulent Dispersion** tab of the **Set Injection Properties** dialog box.

Shared Memory Options
contains settings for the **Shared Memory** option. This group box is only available if **Shared Memory** is selected from the **Methods** list.

Workpile Algorithm
specifies that the discrete phase calculations are to be performed on a shared-memory multiprocessor machine.

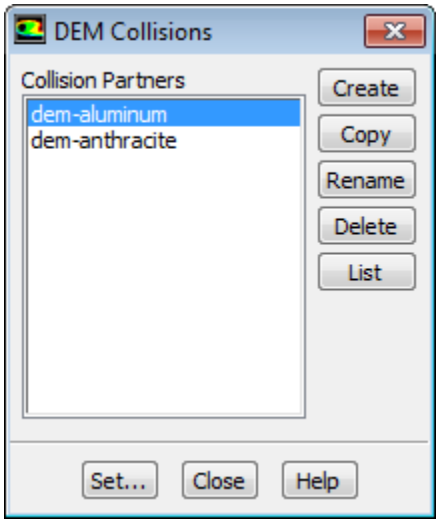
Number of Threads
specifies the number of threads to be used in performing the particle calculations. By default, this parameter is equal to the number of compute nodes you specified for the parallel solver. (This item appears only if **Workpile Algorithm** is enabled.)

Hybrid Options
contains settings for the **Hybrid** option. This group box is only available if **Hybrid** is selected from the **Methods** list.

Use DPM Domain
enables the use of a separate computational domain for particle tracking, which improves load balancing and scalability at the expense of some additional memory overhead.

37.4.40. DEM Collisions Dialog Box

The **DEM Collisions** dialog box allows you to manage the collision partners.



Controls

Collision Partners
contains a list from which you can select one or more collision partners in order to set, create, copy, rename, delete, or list.

Create
creates a new collision partner and opens the Create Collision Partner Dialog Box (p. 2286), in which you can enter the collision partner name.

Copy
creates a new injection with the same properties as the selected collision partner and opens the Copy Collision Partner Dialog Box (p. 2286).

Rename
allows you to change the name of the collision partner. It opens the Copy Collision Partner Dialog Box (p. 2286).

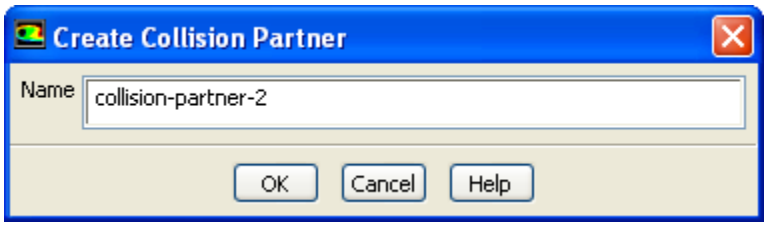
Delete
deletes the collision partner selected in the **Collision Partners** list.

List
lists the collision partners with their corresponding laws.

Set...
opens the DEM Collision Settings Dialog Box (p. 2287) for the collision partner selected in the **Collision Partners** list.

37.4.41. Create Collision Partner Dialog Box

The **Create Collision Partner** dialog box allows you to specify the name of the collision partner.

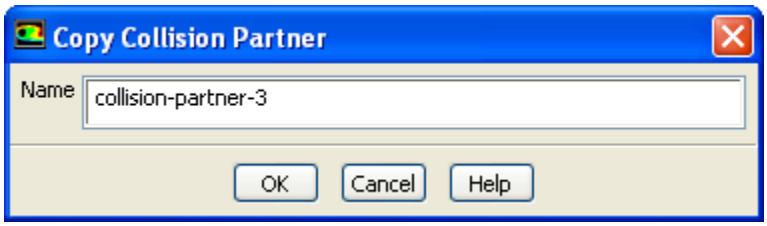


Controls

Name
is the name of the collision partner being created.

37.4.42. Copy Collision Partner Dialog Box

The **Copy Collision Partner** dialog box allows you to copy the selected collision partner.



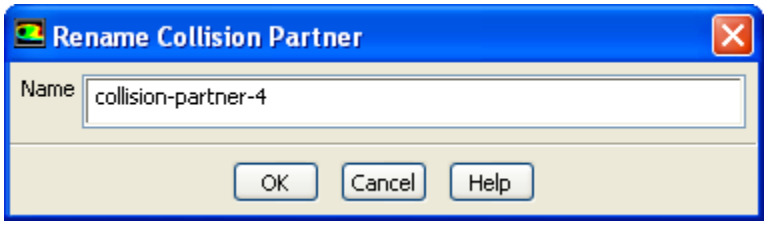
Controls

Name
is the name of the new collision partner copied from the selected partner.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.4.43. Rename Collision Partner Dialog Box

The **Rename Collision Partner** dialog box allows you to rename the selected collision partner.

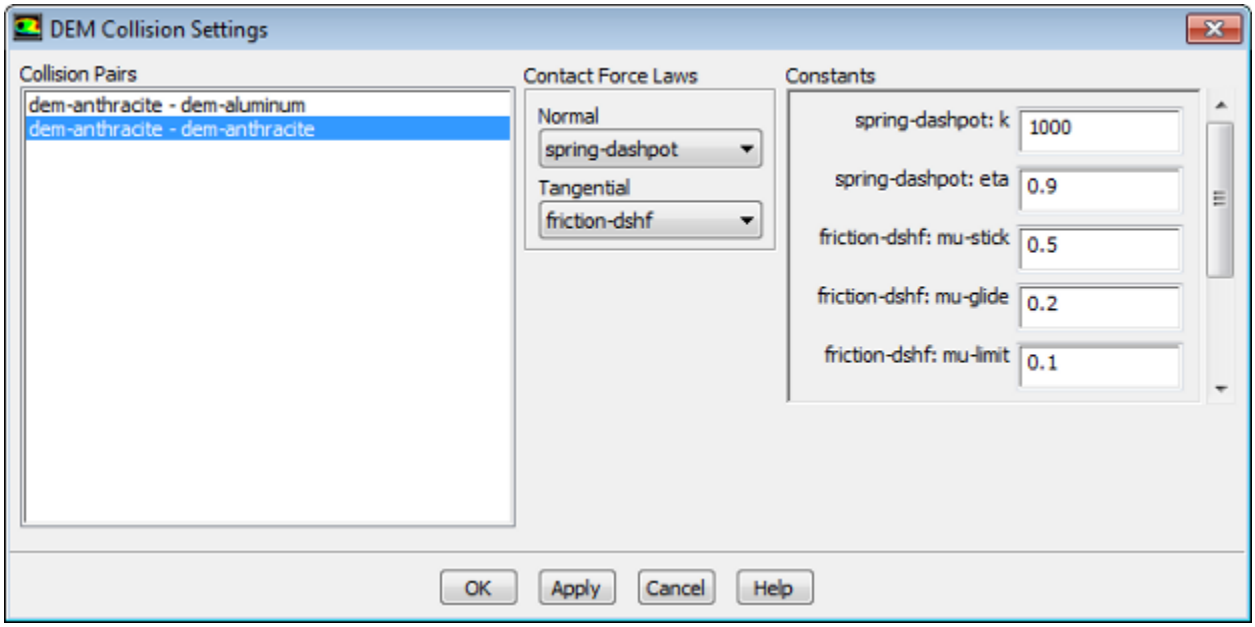


Controls

Name
is the new name of the collision partner selected in the list.

37.4.44. DEM Collision Settings Dialog Box

The **DEM Collision Settings** dialog box allows you to specify the collision laws of the collision partners.



Controls

Collision Pair
contains the list of created collision partners.

Contact Force Laws
is where you will define the collisions laws.

Normal
allows you to choose between **spring**, **spring-dashpot**, **hertzian**, and **hertzian-dashpot**. You can also choose **none** if you do not want to include a contact force.

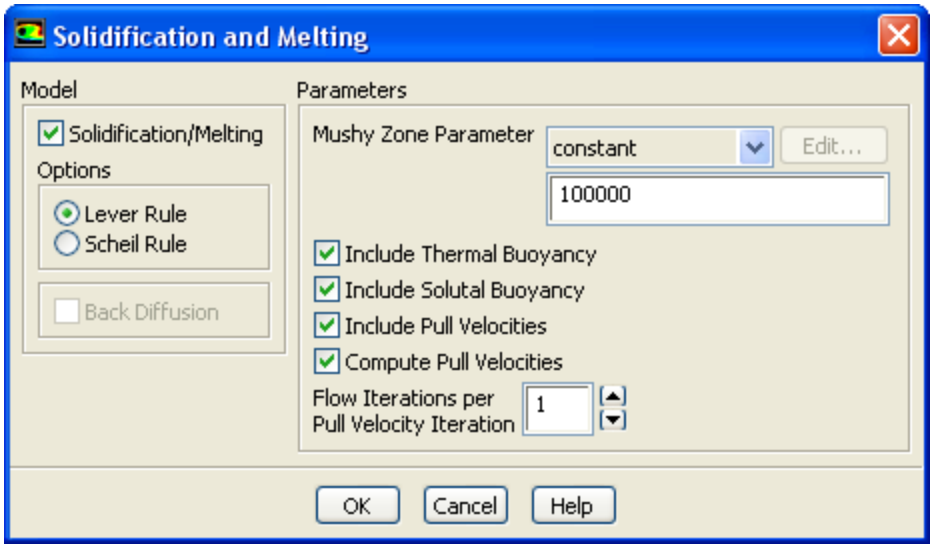
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tangential
allows you to choose between the **friction-dhsf** and **friction-dshf-rolling** force laws. You can also exclude a force law, in which case you would select **none**.

Constants
contains the list of contact force constants. Depending on the contact force law in effect, different constants will be visible.

37.4.45. Solidification and Melting Dialog Box

The **Solidification and Melting** dialog box allows you to set parameters related to the solidification/melting model. See Modeling Solidification and Melting (p. 1527) for details about the items below.



Controls

Model
contains the option for turning on the model.

Solidification/Melting
enables/disables the modeling of solidification and/or melting.

Options
gives you the option of selecting one of two model rules.

Lever Rule
allows you to use the Lever rule (Equation 18.14 in the Theory Guide).

Scheil Rule
allows you to use the Scheil rule (Equation 18.18 in the Theory Guide). If you select **Scheil Rule**, then you can enable **Back Diffusion**. Enter either a **constant** or a user-defined function to specify the value of the **Back Diffusion Parameter**, used in Equation 18.19 to Equation 18.21 in the Theory Guide. Refer to `DEFINE_SOLIDIFICATION_PARAMS` of the Fluent Customization Manual for detailed information about the user-defined function.

Parameters
contains parameters related to the solidification/melting model.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mushy Zone Constant
sets the value of A_{mush} in Equation 18.6 in the Theory Guide.

Include Thermal Buoyancy
includes the gravitational force due to the variation of density with temperature. Refer to Thermal and Solutal Buoyancy in the Theory Guide for more information.

> **Note**
>
> This option is available only when solidification is modeled with species transport.

Include Solutal Buoyancy
includes the gravitational force due to the variation of density with the change in the species composition of the melt. Refer to Thermal and Solutal Buoyancy in the Theory Guide for more information.

> **Note**
>
> This option is available only when solidification is modeled with species transport.

Include Pull Velocities
includes pull velocities in the model, as described in Momentum Equations and Pull Velocity for Continuous Casting in the Theory Guide.

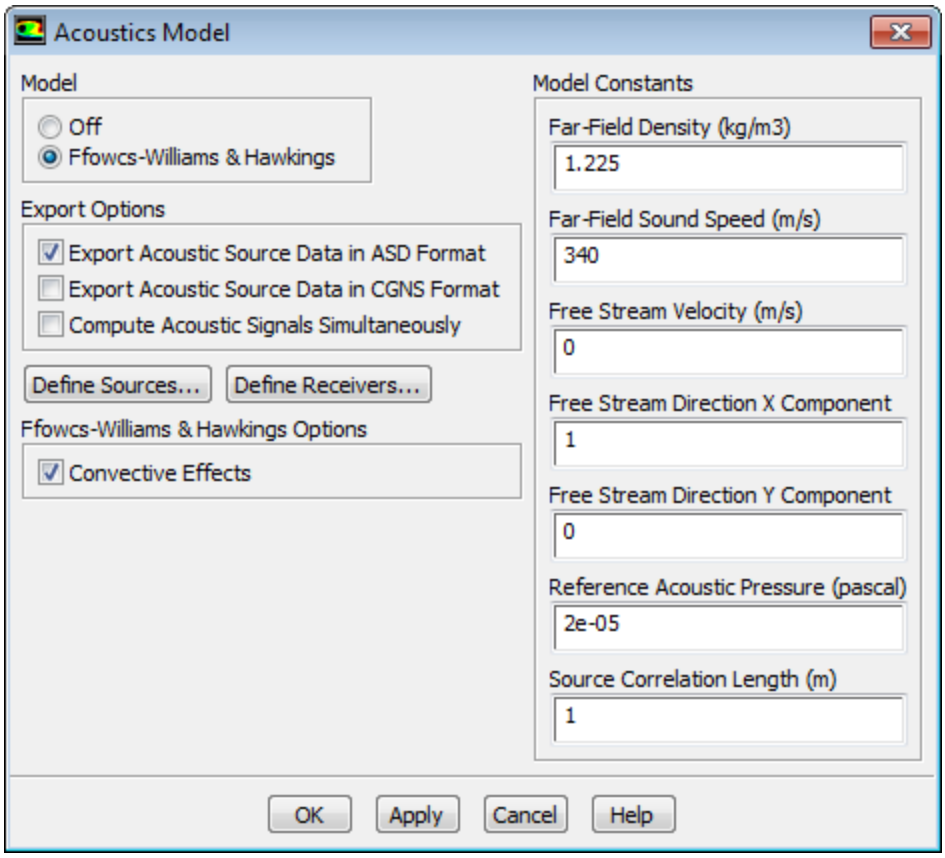
Compute Pull Velocities
enables the calculation of pull velocities based on the specified velocity boundary conditions, as described in Pull Velocity for Continuous Casting in the Theory Guide. This option appears when **Include Pull Velocities** is turned on.

Flow Iterations per Pull Velocity Iteration
specifies the number of times the pull velocity equations will be solved after each iteration of the solver. This option appears when **Compute Pull Velocities** is turned on.

37.4.46. Acoustics Model Dialog Box

The **Acoustics Models** dialog box allows you to set parameters related to the acoustics model. See Using the Ffowcs-Williams and Hawkings Acoustics Model (p. 1207) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Model
contains the option for turning on the acoustics model.

Off
disables the acoustics model.

Ffowcs-Williams & Hawkings
enables the Ffowcs Williams and Hawkings (FW-H) acoustics model. See Postprocessing the FW-H Acoustics Model Data (p. 1219) for details.

Broadband Noise Sources
enables the broadband noise acoustics model. See Using the Broadband Noise Source Models (p. 1231) for details.

Export Options
contains the parameters related to **Ffowcs-Williams & Hawkings** model.

Export Acoustic Source Data in ASD Format
enables/disables the saving of source data files in ASD format.

Export Acoustic Source Data in CGNS Format
enables/disables the saving of source data files in CGNS format.

Compute Acoustics Signals Simultaneously
enables "on-the-fly" calculation of sound. When this option is chosen, the ANSYS Fluent console window will print a message at the end of each time step indicating that the sound pressure signals have been

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

computed (for example, `Extracting sound signals at x receiver locations...`, where `x` is the number of receivers you specified). Enabling this option instructs ANSYS Fluent to compute sound pressure signals at the end of each time step, which will slightly increase the computation time.

Ffowcs-Williams & Hawkings Options
contains the **Convective Effects** option, which should be enabled for all cases dealing with external flows around bodies. When this option is enabled, you will need to specify the proper **Free Stream Velocity** and **Free Stream Direction**.

Define Sources...
opens the Acoustic Sources Dialog Box (p. 2292), which allows you to specify the acoustics sources.

Define Receivers...
opens the Acoustic Receivers Dialog Box (p. 2293), which allows you to specify the location of the receivers.

Model Constants
contains parameters related to the acoustics model.

Far-Field Density
sets the value of the far-field fluid density (ρ_0 in Equation 15.1 in the Theory Guide).

Far-Field Sound Speed
sets the speed of the sound at the far-field (a_0 in Equation 15.5 and Equation 15.6 in the Theory Guide).

Free Stream Velocity
is available when the **Convective Effects** option for the **Ffowcs-Williams & Hawkings** model is enabled.

Free Stream Direction
is available when the **Convective Effects** option for the **Ffowcs-Williams & Hawkings** model is enabled.

Reference Acoustic Pressure
is used to calculate the sound pressure level in dB.

Reference Acoustic Power

is used to non-dimensionalize the acoustic power, giving the sound pressure level in dB (see Fast Fourier Transform (FFT) Postprocessing (p. 1873)). This parameter is equal to 4×10^{-10} W for airborne sound and 10^{-12} W for underwater sound.

Reference Acoustic Intensity

is defined as $P_{ref}^2 / \left(\rho_0 a_0\right)$, although this quantity is not currently used by ANSYS Fluent.

Source Correlation Length
is required when sound is to be computed using a 2D flow result. The FW-H integrals will be evaluated over this length in the depthwise direction using the identical source data.

Number of Realizations
is the number of times the noise source terms will be computed through the generation of stochastic turbulent velocity field.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number of Fourier Modes
is the number of terms in the Fourier summation from which the turbulent velocity field and its derivatives are computed.

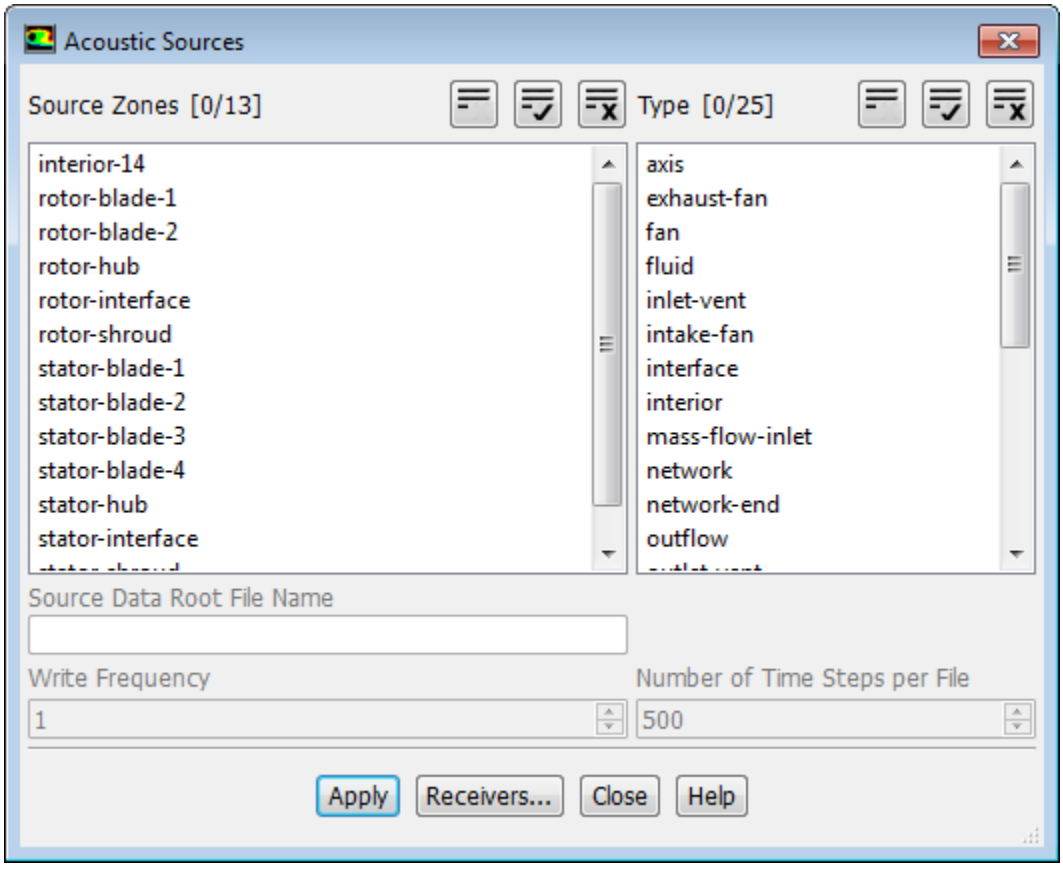
Number of Time Steps Per Revolution
is available only for steady-state cases that have a single moving reference frame. Here you will specify the number of equivalent time steps that it will take for the rotating zone to complete one revolution.

Number of Revolutions
is available only for steady-state cases that have a single moving reference frame. Here you will specify the number of revolutions that will be simulated in the model.

Apply
saves the acoustics model settings.

37.4.47. Acoustic Sources Dialog Box

The **Acoustic Sources** dialog box allows you to specify the acoustics sources. See Using the Ffowcs-Williams and Hawkings Acoustics Model (p. 1207) for details about the items below.



Controls

Source Zones
contains a list of zones from which you can select one or more emission (source) surfaces.

Type
contains a list of all possible types of zones from which you can select one or more types of the available zones. If you select an available type of zone, the relevant zones will then be selected in the **Source Zones**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

list. If you specify any valid **interior** zones as source surfaces, the Interior Cell Zone Selection Dialog Box (p. 2294) will appear.

Source Data Root File Name
is used to give the names of the source data files (for example, `acoustic_example_xxxx.asd`, where `xxxx` is the global time-step index of the transient solution) and an index file (for example, `acoustic_example.index`) that will store the information associated with the source data.

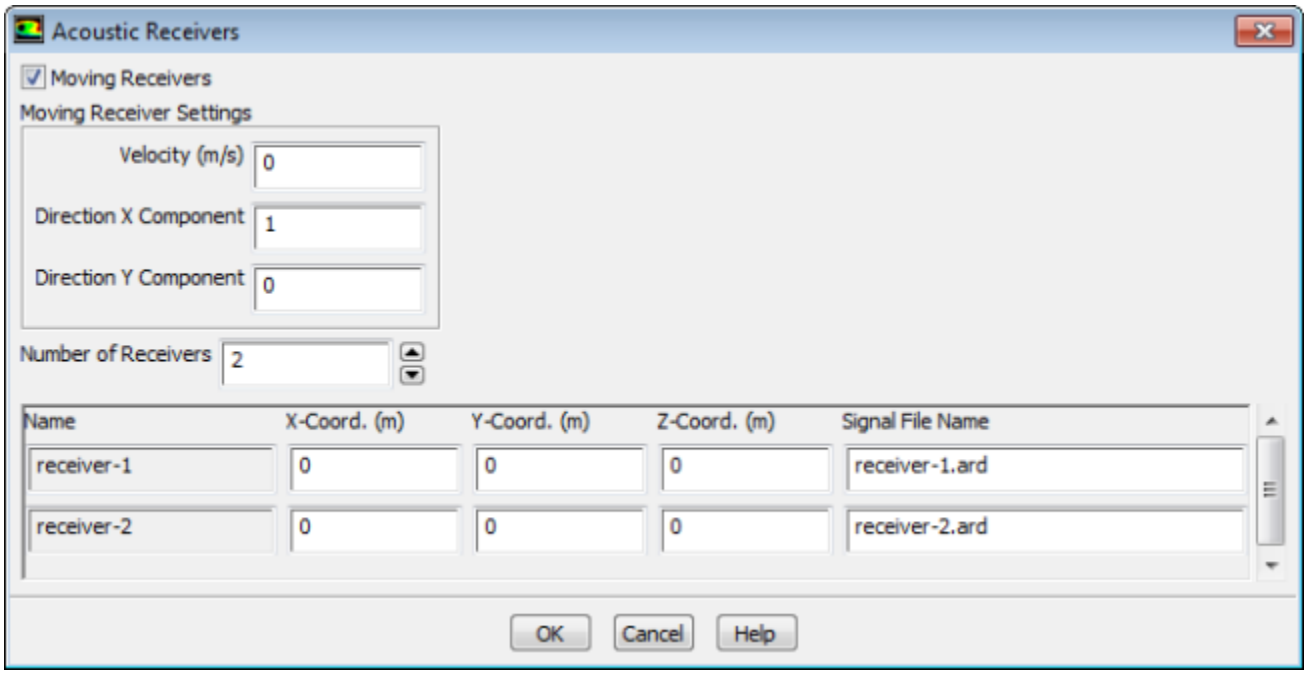
Write Frequency
allows you to control how often the source data will be written. This will enable you to save disk space if the time-step size used in the transient flow simulation is smaller than necessary to resolve the sound frequency you are attempting to predict.

Number of Time Steps per File
allows you to write the source data to multiple files, each containing the source data for a number of time steps specified by you.

Receivers...
opens the Acoustic Receivers Dialog Box (p. 2293), which allows you to specify the location of the receivers.

37.4.48. Acoustic Receivers Dialog Box

The **Acoustic Receivers** dialog box allows you to specify the location of the acoustic receivers. See Using the Ffowcs-Williams and Hawkings Acoustics Model (p. 1207) for details about the items below.



Controls

Moving Receivers
allows you to specify the locations of the moving receivers.

Moving Receiver Settings
allows you to specify the **Velocity** and the **Direction** of the moving receivers.

Number of Receivers
allows you to specify the total number of receivers for which you want to compute sound.

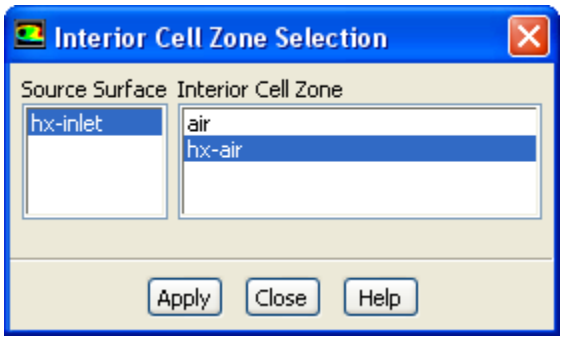
Name
shows the name of the receiver. If you edit the field, the new name will take effect after you click the **OK** button.

X-Coord., Y-Coord., Z-Coord.
specifies the coordinates for each receiver. Note that because ANSYS Fluent's acoustics model is ideally suited for far-field noise prediction, the receiver locations you define should be at a reasonable distance from the sources of sound (that is, the selected acoustic surface), and can fall outside of the computational domain if needed.

Signal File Name
specifies the name of the file used to store sound pressure signals for the corresponding receivers. By default, the files will be named `receiver-1.dat`, `receiver-2.dat`, and so on.

37.4.49. Interior Cell Zone Selection Dialog Box

The **Interior Cell Zone Selection** dialog box allows you to specify the acoustics sources. See Specifying Source Surfaces (p. 1214) for details about the items below.



Controls

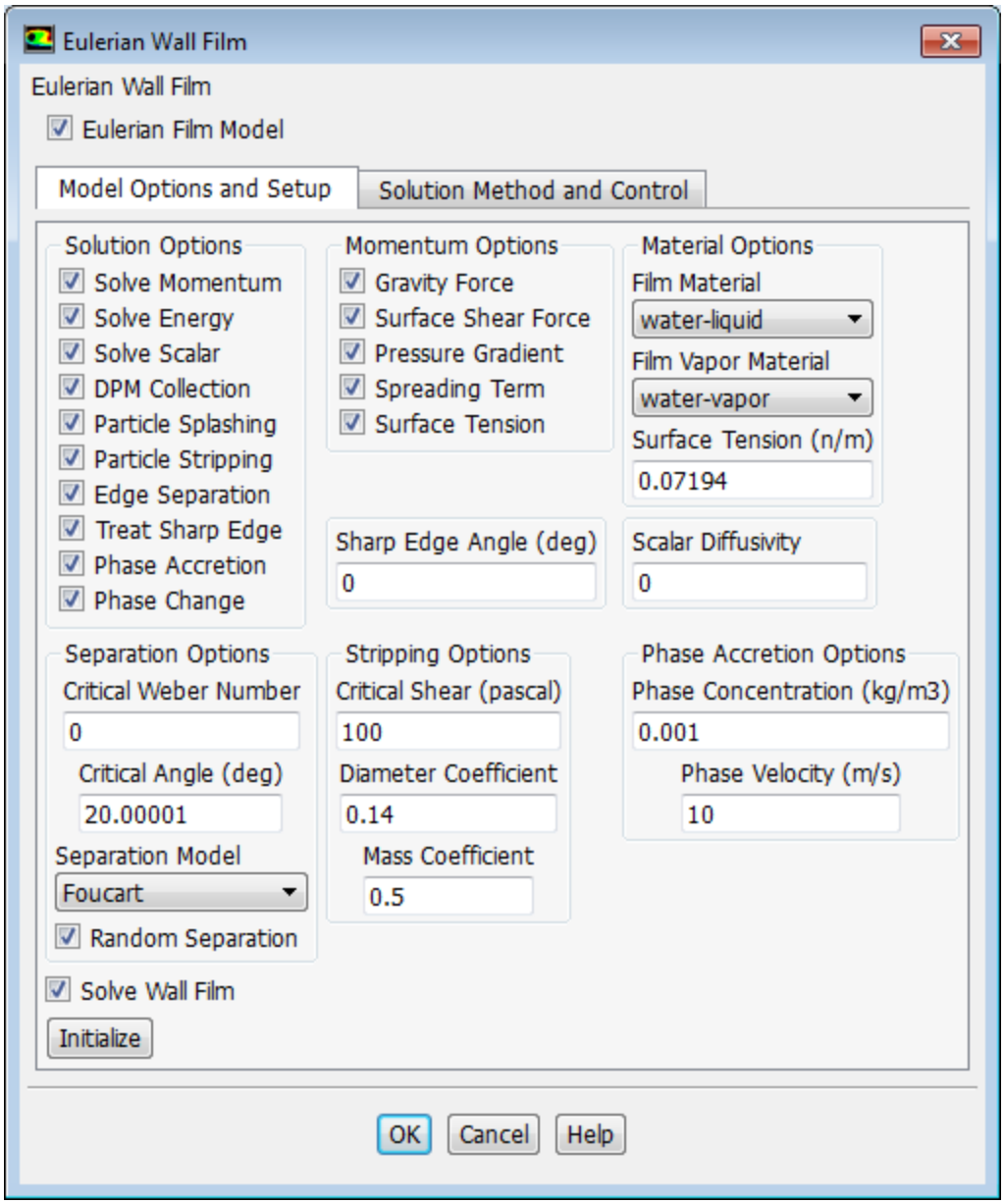
Source Surface
displays the selected interior surface.

Interior Cell Zone
contains a list of two interior cell zones, from which you will select the zone that is occupied by the quadrupole sources.

37.4.50. Eulerian Wall Film Dialog Box

The **Eulerian Wall Film** dialog box allows you to set various model and solution method controls for the Eulerian Wall Film model. For more information, see Modeling Eulerian Wall Films (p. 1535).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Eulerian Wall Film

enables/disables the use of the Eulerian Wall Film model in the calculations.

Model Options and Setup

contains controls for specific solution, discrete phase model (DPM), and material options for the Eulerian Wall Film model.

Solution Options

includes options for enabling/disabling the momentum equation and DPM calculations.

Solve Momentum

allows the momentum equation (see Equation 19.2 in the *Fluent Theory Guide*) to be solved.

Solve Energy

allows the energy equation (see Equation 19.3 in the *Fluent Theory Guide*) to be solved.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solve Scalar
allows the passive scalar equation (see Equation 19.36 in the *Fluent Theory Guide*) to be solved. Once you enable this option, you can set **Scalar Diffusivity**.

DPM Collection
allows for the effect of discrete particle streams or discrete particles hitting a face on a wall boundary that are then absorbed into the film. Once enabled, the **Particle Splashing**, **Particle Stripping**, and **Edge Separation** options become available (see DPM Collection in the Theory Guide). In addition, you can also set **Impingement Parameters** in the Wall Dialog Box (p. 2456).

Particle Splashing
(available when **DPM Collection** is enabled) enables particle splashing globally. You can specify splashing options on selected film wall boundaries in the Wall Dialog Box (p. 2456).

Particle Stripping
(available when **DPM Collection** is enabled) allows you to set the **Critical Shear**, **Diameter Coefficient**, and **Mass Coefficient**, available under **Stripping Options**.

Edge Separation
(available when **DPM Collection** is enabled) allows you to set the **Critical Weber Number**, **Critical Angle**, and **Separation Model**, available under **Separation Options**.

Treat Sharp Edge
allows you to set the **Sharp Edge Angle** when handling the effects of sharp edges and the wall film.

Phase Accretion
(available when the Eulerian Multiphase or Mixture (with Slip Velocity) model is enabled) allows for the interaction of the wall film with a secondary phase in a multiphase flow. Once enabled, the **Phase Concentration** and **Phase Velocity** can be set under **Phase Accretion Options** (see Secondary Phase Accretion in the Theory Guide).

Phase Change
allows you to account for phase changes between the film material (liquid) and the gas species (vapor). When this option is selected, you can specify the phase change parameters on the selected film wall boundaries in the **Wall Film** tab of the Wall Dialog Box (p. 2456). (See Coupling of Wall Film with Mixture Species Transport in the *Fluent Theory Guide*).

Momentum Options
are available when the **Solve Momentum** option is enabled.

Gravity Force
is the second term on the right hand side of Equation 19.2, and is responsible for accelerating the film in the direction of gravity component that is parallel to the wall.

Surface Shear Force
is the third term on the right hand side of Equation 19.2, and is responsible for accelerating the film in the direction of the external flow.

Pressure Gradient
is the first term on the right hand side of Equation 19.2, and is the term accelerating the film in the direction opposing the gradient in external pressure. For example, if a film on the surface of a wing is being modeled and there is a high pressure region at the leading edge with low pressure on the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

top of the wing, this term will tend to move a uniform film towards the low pressure region on the top of the wing.

Spreading Term
is the P_h term on the right hand side of Equation 19.2, and is responsible for spreading. This term will accelerate the flow in the direction opposing the gradient in height, moving the film towards regions of lower thickness. This term becomes available only when both **Pressure Gradient** and **Gravity Term** are selected.

Surface Tension
is the P_σ term on the right hand side of Equation 19.2. This term becomes available only when **Pressure Gradient** is selected.

Sharp Edge Angle
allows you to account for sharp edges alongside the wall film, when the **Treat Sharp Edge** option is enabled.

Material Options
allows you to apply material properties to the wall film model.

Film Material
is the material assigned to the wall film.

Film Vapor Material
(available only when **Phase Change** is selected) is the material assigned to the wall film vapor.

Surface Tension
is the surface tension for the designated wall film material.

Separation Options
(available when **DPM Collection** and **Edge Separation** are enabled) allows you to set DPM particle separation options (see Film Separation in the Theory Guide).

Critical Weber Number
is the critical value for the Weber number.

Separation Angle
is the critical angle, θ, that separation occurs.

Separation Model
allows you to specify one of three different models to calculate the number and diameter of the shed particle stream at an edge once separation occurs. Available models include: **Foucart** (the default, see Foucart Separation in the Theory Guide), **O'Rourke** (see O'Rourke Separation), and **Friedrich** (see Friedrich Separation).

Random Separation
(available when **DPM Collection** and **Edge Separation** are enabled) allows the random selection of the locations at which the newly spawn particles are injected (due to film separation) along the edge at which the separation takes place.

Stripping Options
(available when **DPM Collection** and **Particle Stripping** are enabled) allows you to set DPM particle stripping options (see Film Stripping in the Theory Guide).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Critical Shear
is the critical value of the shear rate on the face where liquid film exists, which, when exceeded, causes mass to be taken from the film.

Diameter Coefficient
is the value of F used in Equation 19.20 in the Theory Guide.

Mass Coefficient
is the value of C used in Equation 19.21 in the Theory Guide.

Splashing Options
(available when **DPM Collection** and **Particle Splashing** are enabled) allows you to set DPM particle splashing options (see Film Sub-Models in the Theory Guide).

Splashed Particles
is the number of splashed discrete particles.

Phase Accretion Options
(available when **Phase Accretion** is enabled) allows you to set secondary phase accretion options (see Secondary Phase Accretion in the Theory Guide).

Phase Concentration
is the value of C_d used in Equation 19.24 in the Theory Guide.

Phase Velocity
is the value of V_d used in Equation 19.24 in the Theory Guide.

Solve Wall Film
allows you to skip the wall film solution during the gas phase solution, but keep the variables and setup active.

Initialize
allows you to initialize the wall film variables and prepare the solver for the solution procedure. The film cannot be solved without first initializing the wall film model.

Solution Method and Control
contains controls for temporal or spatial discretization as well as wall film thickness for the Eulerian Wall Film model.

Discretization
allows you to set the temporal or spatial discretization methods.

Time
allows you to specify the temporal discretization methods. Available options include: **First Order Explicit**, **First Order Implicit**, or **Second Order Implicit**.

Continuity
allows you to specify the spatial discretization methods for continuity. Available options include: **First Order Upwind**, or **Second Order Upwind**.

Momentum
allows you to specify the spatial discretization methods for momentum. Available options include: **First Order Upwind**, or **Second Order Upwind**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Energy
allows you to specify the spatial discretization methods for momentum. Available options include: **First Order Upwind**, or **Second Order Upwind**.

Thickness Control
allows you to specify **Maximum Thickness**.

Time Marching and Time Step Control
allows you to set the number of sub-steps that the film model will take between time steps from the main solver. For example, if the flow time step is 0.001 seconds and the number of time steps is set to 10 in the film model, the film model will take 10 time steps of 0.0001 so that the film time and the flow time match at the end of the solution step. For implicit time stepping, the film model will take 5 additional sub iterations per implicit step, stopping the sub-step when the film residual drops below the value set in the **Sub-Iteration Stop** option.

Adaptive Time Stepping
(steady-state flow) enables the adaptive time stepping method to determine the film time step (Δt) using the maximum Courant number and the initial time step (see Steady Flow in the Theory Guide).

Max. Courant Number
is the maximum value of the Courant number used to compute the film time step.

Initial Time Step
is an initial value for the time step used to compute the film time step.

Time-Step
(steady-state flow; default) is the user-specified size of the time step, Δt (see Steady Flow in the Theory Guide).

Sub-Time Steps
(transient flow) is N_{film} used in Equation 19.50 in the Theory Guide.

Sub-Time Report Interval
(transient flow) specifies the number of iterations between the wall film solution reports.

Sub-Iterations
(available when either **First Order Implicit** or **Second Order Implicit** are enabled for **Time** under **Discretization**) is the number of additional sub iterations per implicit step.

Sub-Iteration Stop
(available when either **First Order Implicit** or **Second Order Implicit** are enabled for **Time** under **Discretization**) is the point at which the sub iterations stop (when the film residual drops below this value).

Sub-Iter Report Interval
(implicit time discretization calculation) specifies the number of sub-iterations between the wall film solution reports.

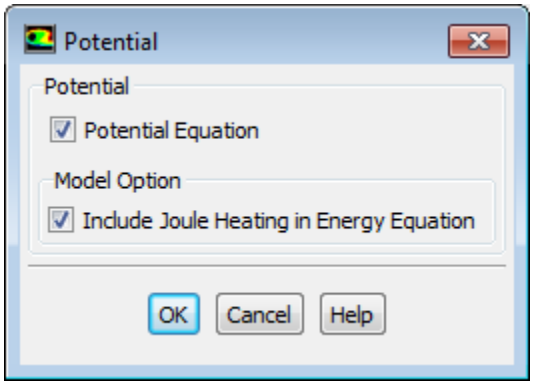
DPM Control
(available when **DPM Collection** is enabled in the **Model Options and Setup** tab) allows you to specify DPM-specific discretization.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Film Steps per DPM step
allows you to set how often the DPM phase is calculated for the film.

37.4.51. Potential Dialog Box

The **Potential** dialog box allows you to set parameters related to the electric potential field in your model. See Using the Electric Potential Model (p. 1547) for details.



Controls

Potential
contains inputs related to modeling of the electric potential field.

Potential Equation
enables/disables the simulation of the electric potential field in the model.

Model Option
allows you to enable/disable the **Include Joule Heating in Energy Equation** option. This option appears only if the energy equation is enabled.

37.5. Materials Task Page

The **Materials** task page allows you to set properties for any fluid or solid (or mixture, if applicable) materials in your ANSYS Fluent simulation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Materials
contains a listing of available fluid or solid (or mixture, if applicable) materials.

Create/Edit...
displays the Create/Edit Materials Dialog Box (p. 2302) for the selected item in the **Materials** list.

Delete
removes the selected material from the **Materials** list,

For additional information, see the following sections:

37.5.1. Create/Edit Materials Dialog Box
37.5.2. Fluent Database Materials Dialog Box
37.5.3. Open Database Dialog Box
37.5.4. User-Defined Database Materials Dialog Box
37.5.5. Copy Case Material Dialog Box
37.5.6. Material Properties Dialog Box
37.5.7. Edit Property Methods Dialog Box
37.5.8. New Material Name Dialog Box
37.5.9. Polynomial Profile Dialog Box
37.5.10. Piecewise-Linear Profile Dialog Box
37.5.11. Piecewise-Polynomial Profile Dialog Box
37.5.12. Convection/Diffusion Model Dialog Box
37.5.13. Compressible Liquid Dialog Box
37.5.14. User-Defined Functions Dialog Box

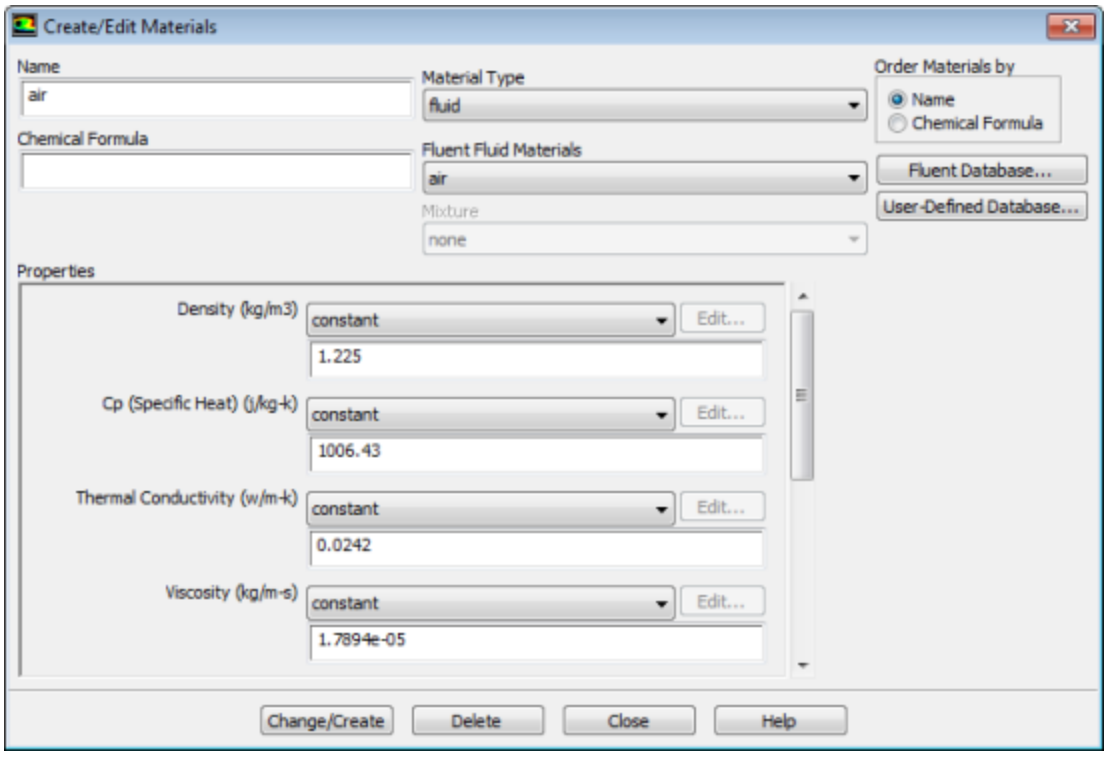
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.5.15. Sutherland Law Dialog Box
37.5.16. Power Law Dialog Box
37.5.17. Non-Newtonian Power Law Dialog Box
37.5.18. Carreau Model Dialog Box
37.5.19. Cross Model Dialog Box
37.5.20. Herschel-Bulkley Dialog Box
37.5.21. Biaxial Conductivity Dialog Box
37.5.22. Cylindrical Orthotropic Conductivity Dialog Box
37.5.23. Orthotropic Conductivity Dialog Box
37.5.24. Anisotropic Conductivity - Principal Components Dialog Box
37.5.25. Anisotropic Conductivity Dialog Box
37.5.26. Species Dialog Box
37.5.27. Reactions Dialog Box
37.5.28. Backward Reaction Parameters Dialog Box
37.5.29. Third-Body Efficiency Dialog Box
37.5.30. Pressure-Dependent Reaction Dialog Box
37.5.31. Coverage-Dependent Reaction Dialog Box
37.5.32. Reference Mass Fractions Dialog Box
37.5.33. Reaction Mechanisms Dialog Box
37.5.34. Site Parameters Dialog Box
37.5.35. Mass Diffusion Coefficients Dialog Box
37.5.36. Thermal Diffusion Coefficients Dialog Box
37.5.37. UDS Diffusion Coefficients Dialog Box
37.5.38. WSGGM User Specified Dialog Box
37.5.39. Gray-Band Absorption Coefficient Dialog Box
37.5.40. Delta-Eddington Scattering Function Dialog Box
37.5.41. Gray-Band Refractive Index Dialog Box
37.5.42. Single Rate Devolatilization Dialog Box
37.5.43. Two Competing Rates Model Dialog Box
37.5.44. CPD Model Dialog Box
37.5.45. Kinetics/Diffusion-Limited Combustion Model Dialog Box
37.5.46. Intrinsic Combustion Model Dialog Box
37.5.47. Multiple Surface Reactions Dialog Box
37.5.48. Edit Material Dialog Box

37.5.1. Create/Edit Materials Dialog Box

The **Create/Edit Materials** dialog box is used to create and modify materials. Materials can be downloaded from the global database or defined locally. See Physical Properties (p. 459) for details about defining material properties. Using the **Create/Edit Materials** Dialog Box (p. 461) describes how to use the dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

shows the name of the material. If you edit this field, the new name will take effect when you click **Change/Create**.

Chemical Formula

displays the chemical formula for the material. You should generally not edit this field unless you are creating a material from scratch.

Material Type

is a drop-down list containing all of the available material types. By default, **fluid** and **solid** will be the only choices. If you are modeling species transport/combustion, **mixture** will also be available. For problems in which you have defined discrete-phase injections, **inert-particle**, **droplet-particle**, and/or **combusting-particle** will also appear.

Fluent Fluid Materials

allows you to choose the fluid material for which you want to modify properties. This option is available when **fluid** is selected in the **Material Type** drop-down list.

Fluent Solid Materials

allows you to choose the solid material for which you want to modify properties. This option is available when **solid** is selected in the **Material Type** drop-down list.

Fluent Mixture Materials

allows you to choose the mixture material for which you want to modify properties. This option is available when **mixture** is selected in the **Material Type** drop-down list.

Fluent Droplet Particle Materials

allows you to choose the droplet-particle for which you want to modify properties. This option is available when **droplet-particle** is selected in the **Material Type** drop-down list.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Order Materials by
allows you to order the materials in the **Materials** list alphabetically by **Name** or alphabetically by **Chemical Formula**.

Fluent Database...
opens the Fluent Database Materials Dialog Box (p. 2312), from where you can copy materials from the global database into the current solver.

User-Defined Database...
opens the Open Database Dialog Box (p. 2313), where you can specify the user-defined database to be used.

Properties
contains input fields for the material properties that are required for the active physical models.

Density
sets the material density. You may set a constant value, or select one of the other methods from the drop-down list above the real number field. See Density (p. 479) for instructions on setting density.

Cp (Specific Heat)
sets the constant-pressure specific heat of the material. You may set a constant value, or select one of the other methods from the drop-down list above the real number field. See Specific Heat Capacity (p. 513) for instructions on setting specific heat.

Thermal Conductivity
sets the thermal conductivity of the material. You may set a constant value, or select one of the other methods from the drop-down list above the real number field. See Thermal Conductivity (p. 497) for instructions on setting thermal conductivity.

Viscosity
sets the viscosity of the material. You may set a constant value, or select one of the other methods from the drop-down list above the real number field. See Viscosity (p. 487) for instructions on setting viscosity.

Molecular Weight
sets the molecular weight of the material. It is used to derive the gas constant of the material.

Standard State Enthalpy
specifies the formation enthalpy of a fluid material for a reacting flow. See Standard State Enthalpies (p. 528) for details.

Standard State Entropy
specifies the standard state entropy of a fluid material for a reacting flow. This input is used only if the fluid material is involved in a reversible reaction. See Standard State Entropies (p. 528) for details.

Reference Temperature
specifies the reference temperature for the **Heat of Formation**.

L-J Characteristic Length
specifies the kinetic theory parameter σ for a fluid material. See Kinetic Theory Parameters (p. 529) for details.

L-J Energy Parameter
specifies the kinetic theory parameter ε / k for a fluid material. See Kinetic Theory Parameters (p. 529) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Absorption Coefficient

specifies the absorption coefficient a for radiation heat transfer. See Radiation Properties (p. 515) for details. If you choose the **wsggm-user-specified** option from the drop-down list next to **Absorption Coefficient**, the WSGGM User Specified Dialog Box (p. 2348) will open. If you choose the **user-defined-wsggm** option from the drop-down list next to **Absorption Coefficient**, the User-Defined Functions Dialog Box (p. 2321) will open.

Scattering Coefficient

specifies the scattering coefficient σ_s for radiation heat transfer (only for the P-1, Rosseland, DO, or MC radiation models). See Radiation Properties (p. 515) for details.

Scattering Phase Function

specifies an **isotropic** (by default) or **linear-anisotropic** scattering function. If you are using the DO model, **delta-eddington** and **user-defined** scattering functions are also available. See Radiation Properties (p. 515) for details. If you choose **delta-eddington**, the Delta-Eddington Scattering Function Dialog Box (p. 2349) will open.

Refractive Index

specifies the refractive index for the material. It is used only when semi-transparent media are modeled with the DO radiation model.

Mixture Species

specifies the names of the species that comprise a mixture material. To check or modify these names, click the **Edit...** button to open the Species Dialog Box (p. 2332). This property appears only for mixture materials.

Reaction

displays the reaction mechanism being used when you are modeling finite-rate reactions. **finite-rate** appears if **LaminarFinite-Rate** or **Eddy-Dissipation Concept** is selected in the Species Model Dialog Box (p. 2216), **eddy-dissipation** appears if **Eddy-Dissipation** is selected, and **finite-rate/eddy-dissipation** appears if **Finite-Rate/Eddy-Dissipation** is selected.

Click **Edit...** to open the Reactions Dialog Box (p. 2334).

Mechanism

allows you to enable different reactions selectively in different geometrical zones. Click the **Edit** button to open the Reaction Mechanisms Dialog Box (p. 2342). See Defining Zone-Based Reaction Mechanisms (p. 984) for details.

Mass Diffusivity

contains a drop-down list of available methods for specifying the diffusion coefficients for the species in a mixture material. If you select **constant-dilute-appx**, you will enter a constant value in the field below. If you select **dilute-approx** or **multicomponent**, the Mass Diffusion Coefficients Dialog Box (p. 2344) will open, and you can specify the coefficients there. If you select **kinetic-theory**, you will need to specify the kinetic theory parameters for the individual fluid materials (species) that comprise the mixture. See Mass Diffusion Coefficients (p. 519) for details about specifying mass diffusivity.

Thermal Diffusion Coefficient

contains a drop-down list of available methods for specifying the thermal diffusion coefficients for the species in a mixture material. If you select **kinetic-theory**, you will need to specify the kinetic theory materials (species) that comprise the mixture. If you select **specified**, the Thermal Diffusion Coefficients Dialog Box (p. 2346) will open, and you can specify the coefficients there. See Thermal Diffusion Coefficient Inputs (p. 523) for details about specifying thermal diffusion coefficients.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Density of Unburnt Reactants
sets the density (ρ_u in Equation 9.68 in the Theory Guide) of the unburnt products.

Temperature of Unburnt Reactants
sets the temperature (T_u in Equation 9.68 in the Theory Guide) of the unburnt products.

Adiabatic Temperature of Burnt Products
(only for adiabatic premixed combustion models) specifies the value of the burnt products under adiabatic conditions, T_{ad} in Equation 9.65 in the Theory Guide.

Unburnt Thermal Diffusivity
specifies the thermal diffusivity (α in Equation 9.9 in the Theory Guide) for use with the premixed combustion model. See Modeling Premixed Combustion (p. 1089) for details.

Laminar Flame Speed
specifies the value of U_l in Equation 9.9 in the Theory Guide.

Laminar Flame Thickness
specifies the thickness of the flame for which you have a choice of **constant**, **user-defined**, or **diffusivity-over-flame-speed**.

Critical Rate of Strain
specifies the value of g_{cr} in Equation 9.17 in the Theory Guide.

Heat of Combustion
(only for non-adiabatic premixed combustion models) specifies the value of H_{comb} in Equation 9.67 in the Theory Guide.

Unburnt Fuel Mass Fraction
(only for non-adiabatic premixed combustion models) specifies the value of Y_{fuel} in Equation 9.67 in the Theory Guide.

Thermal Expansion Coefficient
specifies the thermal expansion coefficient (β in Equation 12.3 (p. 833)) for use with the Boussinesq approximation. See Steps in Solving Buoyancy-Driven Flow Problems (p. 834) for details.

Droplet Surface Tension
specifies the value of the droplet surface tension (σ in Equation 16.333 in the Theory Guide).

Latent Heat
is the latent heat of vaporization, h_{fg}, required for phase change from an evaporating liquid droplet or for the evolution of volatiles from a combusting particle. See Setting Material Properties for the Discrete Phase (p. 1313) for details.

Latent Heat at NBP
is the latent heat of vaporization at the normal boiling point, available for droplet-particles. See Using the Cubic Equation of State Models with the Lagrangian Dispersed Phase Models (p. 548) for details.

Normal Boiling Point (NBP)
is given by Equation 7.117 (p. 549). See Using the Cubic Equation of State Models with the Lagrangian Dispersed Phase Models (p. 548) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Thermophoretic Coefficient

specifies the thermophoretic coefficient ($D_{T,p}$ in Equation 16.11 in the Theory Guide), and appears when the thermophoretic force is included in the discrete phase calculation.

Vaporization Temperature

is the temperature, T_{vap}, at which the calculation of vaporization from a liquid droplet or devolatilization from a combusting particle is initiated by ANSYS Fluent. See Setting Material Properties for the Discrete Phase (p. 1313) for details.

Boiling Point

is the temperature, T_{bp}, at which the calculation of the boiling rate equation is initiated by ANSYS Fluent. See Setting Material Properties for the Discrete Phase (p. 1313) for details.

Vapor-Particle-Equilibrium

is the selected approach for the calculation of the vapor concentration of the components at the surface. This can be Raoult's law (Equation 16.191 in the Theory Guide), the Peng-Robinson real gas model (Equation 16.199 in the Theory Guide), or a user-defined function that provides this value.

Volatile Component Fraction

(f_{v0}) is the fraction of a droplet particle that may vaporize via Laws 2 and/or 3 (Droplet Vaporization (Law 2) in the Theory Guide). For combusting particles, it is the fraction of volatiles that may be evolved via Law 4 (Devolatilization (Law 4) in the Theory Guide). See Setting Material Properties for the Discrete Phase (p. 1313) for details.

Binary Diffusivity

is the mass diffusion coefficient, $D_{i,m}$, used in the vaporization law, Law 2. This input is also used to define the mass diffusion of the oxidizing species to the surface of a combusting particle, $D_{i,m}$. See Setting Material Properties for the Discrete Phase (p. 1313) for details.

Diffusivity Reference Pressure

is the reference pressure P_{ref} for the pressure dependent binary diffusivity in Equation 16.110 in the *Fluent Theory Guide*.

Saturation Vapor Pressure

is the saturated vapor pressure, P_{sat}, defined as a function of temperature, which is used in the vaporization law, Law 2. See Setting Material Properties for the Discrete Phase (p. 1313) for details.

Heat of Pyrolysis

is the heat of the instantaneous pyrolysis reaction, h_{pyrol}, that the evaporating/boiling species may undergo when released to the continuous phase. The heat of pyrolysis should be entered as a positive number for exothermic reaction and as a negative number for endothermic reaction. The default value of zero implies that the heat of pyrolysis is not considered. See Setting Material Properties for the Discrete Phase (p. 1313) for details.

Composition Averaging Coefficient

is the coefficient α_Y in Equation 16.201 in the *Fluent Theory Guide*. To assume bulk gas composition for the physical properties in the particle vaporization and heating rates equations, select `none` from the **Composition Averaging Coefficient** drop-down in the **Create/Edit Materials** dialog box, under **Properties**. To enable property averaging, select `constant` and enter a value between 0 and 1 for

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the **Composition Averaging Coefficient**. $\alpha_Y = 1$ corresponds to bulk gas mixture. For $\alpha_Y = 0$, the composition reverts to the particle surface composition.

Important

When you define the **Composition Averaging Coefficient** with a `constant` value other than 1, you must also specify **Specific Heat**, **Viscosity** and **Thermal Conductivity** for the evaporating species (in the **Create/Edit Materials** dialog box, under **Properties**, for the corresponding fluid material of the continuous phase mixture).

The default and recommended value for the averaging coefficient is 1/3. However, to improve the simulation results, you must provide accurate temperature-dependent data for the vapor material.

Temperature Averaging Coefficient

is the coefficient α_T in Equation 16.200 in the *Fluent Theory Guide*. To assume bulk gas temperature for the physical properties in the particle vaporization and heating rates equations, select `none` from the **Temperature Averaging Coefficient** drop-down in the **Create/Edit Materials** dialog box, under **Properties**. To enable property averaging, select `constant` and enter a value between 0 and 1 for the **Temperature Averaging Coefficient**. $\alpha_T = 1$ corresponds to bulk gas conditions. For $\alpha_T = 0$, the temperature reverts to the particle surface temperature.

Important

When you define the **Temperature Averaging Coefficient** with a `constant` value other than 1, you must also specify **Specific Heat**, **Viscosity** and **Thermal Conductivity** for the evaporating species (in the **Create/Edit Materials** dialog box, under **Properties**, for the corresponding fluid material of the continuous phase mixture).

The default and recommended value for the averaging coefficient is 1/3. However, to improve the simulation results, you must provide accurate temperature-dependent data for the evaporating vapor material.

Vaporization Model

defines which vaporization model is used for pure droplets (Law 2) and for multicomponent droplets (Law 7). If you want to use the default diffusion controlled model, retain the selection of **diffusion-controlled** from the drop-down list to the right of **Vaporization Model**. This will apply Equation 16.101 in the Theory Guide.

To use the convection/diffusion controlled model for vaporization select **convection/diffusion-controlled** from the drop-down list. Equation 16.106 in the Theory Guide will be applied for the calculation of the vaporization rate, and Equation 16.116 in the Theory Guide will be applied in the particle heat transfer calculations This model is recommended when evaporation rates are high. For slowly evaporating droplets both models are expected to give similar results.

If you have selected the **convection/diffusion-controlled**, you can enable the **Variable Lewis Number Formulation** option in Convection/Diffusion Model Dialog Box (p. 2320) to compute the Spalding heat number from Equation 16.115 in the *Fluent Theory Guide*. If the option is disabled ANSYS Fluent uses Equation 16.116.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Degrees of Freedom

specifies the kinetic theory parameter f, which is the number of nodes of energy storage. This parameter is required only if you are defining specific heat via kinetic theory. See Kinetic Theory Parameters (p. 529) for details.

Particle Emissivity

is the emissivity of particles in your model, ε_p, used to compute radiation heat transfer to the particles when the P-1 or DO radiation model is active and particle radiation interaction is enabled in the Discrete Phase Model Dialog Box (p. 2278). See Setting Material Properties for the Discrete Phase (p. 1313) for details.

Particle Scattering Factor

is the scattering factor, f, due to particles in the P-1 or DO radiation model. Note that this property will appear only if particle radiation interaction is enabled in the Discrete Phase Model Dialog Box (p. 2278). See Setting Material Properties for the Discrete Phase (p. 1313) for details.

Swelling Coefficient

is the coefficient, C_{sw}, which governs the swelling of the coal particle during the devolatilization law, Law 4. A swelling coefficient of unity (the default) implies that the coal particle stays at constant diameter during the devolatilization process. See Setting Material Properties for the Discrete Phase (p. 1313) for details.

Burnout Stoichiometric Ratio

is the stoichiometric requirement, S_b, for the burnout reaction, in terms of mass of oxidant per mass of char in the particle. See Setting Material Properties for the Discrete Phase (p. 1313) for details.

Combustible Fraction

is the mass fraction of char, f_{comb}, in the coal particle (the fraction of the initial combusting particle that will react in the surface reaction) Law 5. See Setting Material Properties for the Discrete Phase (p. 1313) for details.

React. Heat Fraction Absorbed by Solid

is the parameter f_h, which controls the distribution of the heat of reaction between the particle and the continuous phase. The default value of zero implies that the entire heat of reaction is released to the continuous phase. See Setting Material Properties for the Discrete Phase (p. 1313) for details.

Heat of Reaction for Burnout

is the heat released by the surface char combustion reaction, Law 5. This parameter is specified in terms of heat release (for example, Joules) per unit mass of char consumed in the surface reaction. See Setting Material Properties for the Discrete Phase (p. 1313) for details.

Devolatilization Model

defines which version of the devolatilization model, Law 4, is being used. If you want to use the default constant rate devolatilization model, retain the selection of **constant** in the drop-down list to the right of **Devolatilization Model** and enter the rate constant A_0 in the field below the list.

Select **single-rate**, **two-competing-rates**, or **cpd-model** in the drop-down list to choose one of the optional devolatilization models (the single kinetic rate model, two kinetic rates model, or CPD model, as described in Devolatilization (Law 4) in the Theory Guide).

When the single kinetic rate model (**single-rate**) is selected, the Single Rate Devolatilization Dialog Box (p. 2350) will appear; when the two competing rates model (**two-competing-rates**) is selected, the Two Competing Rates Model Dialog Box (p. 2351) will appear; and when the CPD model (**cpd-model**) is selected, the CPD Model Dialog Box (p. 2352) will appear.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See Setting Material Properties for the Discrete Phase (p. 1313) for details.

Combustion Model

defines which version of the surface char combustion law (Law 5) is being used. If you want to use the default diffusion-limited rate model, retain the selection of **diffusion-limited** in the drop-down list. No additional inputs are necessary, because the binary diffusivity defined above will be used in Equation 16.165 in the Theory Guide.

To use the kinetics/diffusion-limited rate model for the surface combustion model, select **kinetics/diffusion-limited** in the drop-down list and enter the parameters in the resulting Kinetics/Diffusion-Limited Combustion Model Dialog Box (p. 2352).

To use the intrinsic model for the surface combustion model, select **intrinsic-model** in the drop-down list and enter the parameters in the resulting Intrinsic Combustion Model Dialog Box (p. 2353).

To use the multiple surface reactions model, select **multiple-surface-reactions** in the drop-down list.

See Setting Material Properties for the Discrete Phase (p. 1313) for details.

Pure Solvent Melting Heat

specifies the latent heat for the melting and solidification model (L in Equation 18.3 in the Theory Guide).

Solidus Temperature

specifies the solidus temperature for the melting and solidification model ($T_{solidus}$ in Equation 18.3 in the Theory Guide). If you are solving for species transport, the solidus temperature is $T_{solidus}$ in Equation 18.8, and you specify the method by which it is calculated: either according to the **mixing-law** (which is based on the parameters of the solutes) or a **user-defined** function.

Liquidus Temperature

specifies the liquidus temperature for the melting and solidification model ($T_{liquidus}$ in Equation 18.3 in the Theory Guide). If you are solving for species transport, the liquidus temperature is T_{liquidus} in Equation 18.9, and you specify the method by which it is calculated: either according to the **mixing-law** (which is based on the parameters of the solutes) or a **user-defined** function.

Pure Solvent Melting Temperature

specifies the melting temperature of pure solvent (T_{melt} in Equation 18.8 and Equation 18.9 in the Theory Guide) for the melting and solidification model when species transport has also been enabled. The solvent is the last species listed under of the mixture material.

Eutectic Temperature

is the lowest alloy melting temperature, which depends on the relative proportions of the mixture composition of the Eutectic species mass fractions.

Slope of Liquidus Line

specifies the slope of the liquidus surface with respect to the concentration of the solute (m_i in Equation 18.8 and Equation 18.9 in the Theory Guide). It is not necessary to specify this value for the solvent. Note that this option is available only for the melting and solidification model when species transport has also been enabled.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Partition Coefficient

specifies the partition coefficient with respect to the concentration of the solute fluid (K_i in Equation 18.8 and Equation 18.9 in the Theory Guide). It is not necessary to specify this value for the solvent. Note that this option is available only for the melting and solidification model when species transport has also been enabled.

Eutectic Mass Fraction

is the mass fraction of a solute in an alloy at which the melting temperature of the alloy is the lowest possible value ($Y_{i,\mathrm{Eut}}$ in Equation 18.10 in the Theory Guide). It is not necessary to specify this value for the solvent. Note that this option is available only for the melting and solidification model when species transport has also been enabled.

Solutal Expansion Coefficient

allows you to specify the coefficient in Equation 18.24 in the Theory Guide for all the species except the last one in the mixture. Note that this option is available only for the melting and solidification model when the **Include Solutal Buoyancy** option is enabled.

Diffusion in Solid

specifies the rate of diffusion in the solid. Note that this option is available only for the melting and solidification model when the Lever rule is selected and the species transport is enabled.

UDS Diffusivity

specifies the diffusion coefficient for a user-defined scalar. This material property is available in the Create/Edit Materials Dialog Box (p. 2302) when you specify one or more user-defined scalars in the User-Defined Scalars Dialog Box (p. 2829). If you select **defined-per-uds**, you will need to specify the diffusion coefficient for each user-defined scalar transport equation in the UDS Diffusion Coefficients Dialog Box (p. 2347).

Electrical Conductivity

sets the material electrical conductivity. You may set a constant value, or select one of the other methods from the drop-down list above the real number field. This material property only becomes available if the electric potential model is enabled.

Charge Number

specifies the charge number for a fluid material for a flow involving electrochemical reactions. The default setting of 0 means that the fluid species is neutral. This material property only becomes available if the electric potential model is enabled.

When you are viewing the database, additional properties may be displayed. However, after you copy the material to the local area, only the properties with relevance to the current problem will be displayed.

Change/Create

changes the properties of a locally stored material or creates a new one in the local area. If no material with the specified **Name** exists locally, ANSYS Fluent will create it. If you have modified the material without changing its name, ANSYS Fluent will simply update the material with your modifications. If you have assigned a new name to the material and a material with this name already exists locally, an error will be indicated; you must then specify a different name or delete the existing material with that name before trying to save the new material.

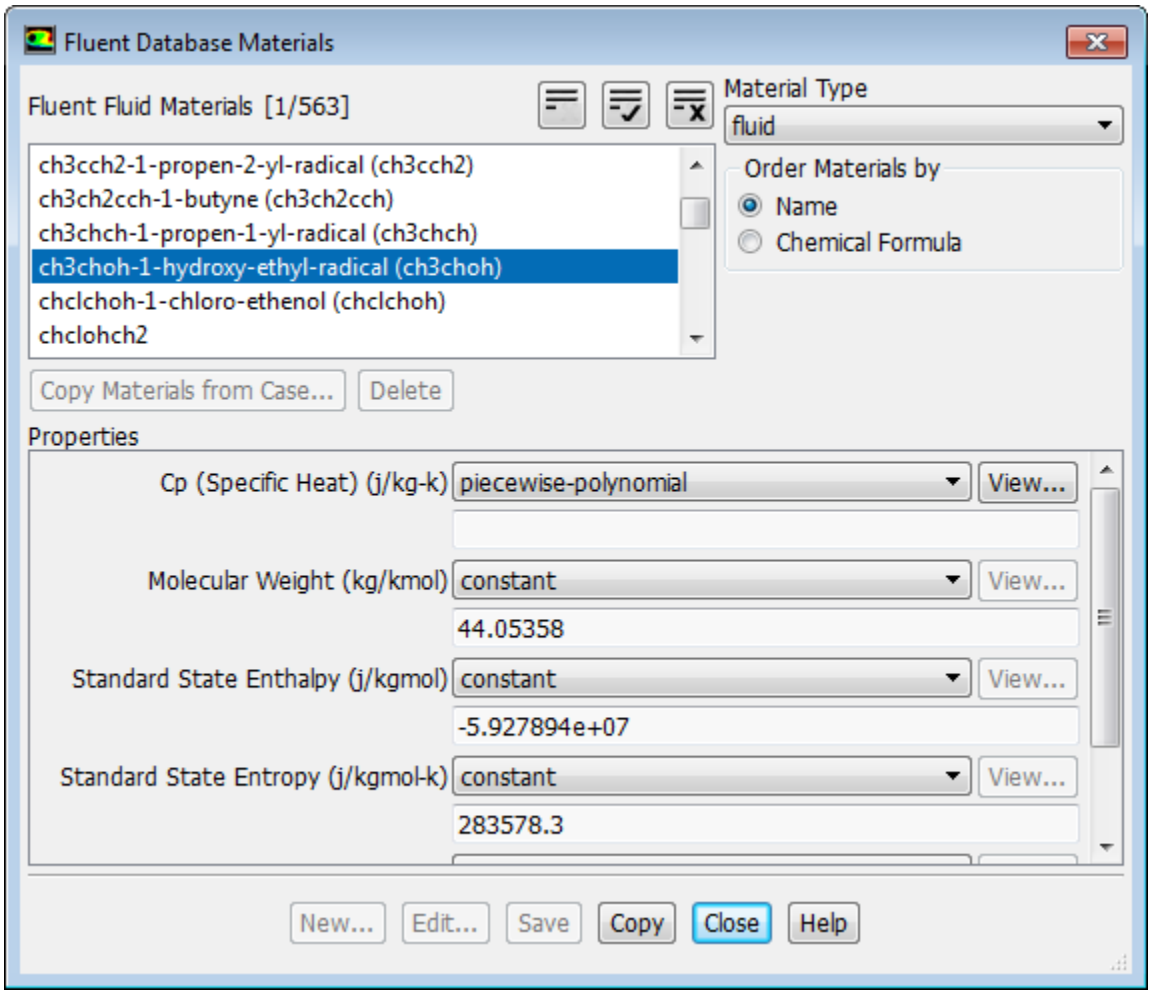
Delete

deletes the currently selected material from the local materials list. It has no effect on the global database.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.5.2. Fluent Database Materials Dialog Box

The **Fluent Database Materials** dialog box is opened by clicking the **Fluent Database...** button in the Create/Edit Materials Dialog Box (p. 2302). In this dialog box you can view the global (site-wide) material properties database and copy materials list in the solver. See Copying Materials from the ANSYS Fluent Database (p. 464) for details.



Controls

Fluent Fluid Materials

contains a list of all materials of the selected **Material Type** that are defined in the database. The name of this list will change depending on the selected material type (for example, **fluid**, **solid**, and so on). You can select one or more of these materials to be copied to the solver.

Material Type

is a drop-down list containing all of the available material types. By default, **fluid** and **solid** will be the only choices. If you are modeling species transport/combustion, **mixture** will also be available. For problems in which you have defined discrete-phase injections, **inert-particle**, **droplet-particle**, and/or **combusting-particle** will also appear.

Order Materials by

allows you to order the materials in the **Materials** list alphabetically by **Name** or alphabetically by **Chemical Formula**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Copy Materials from Case...
opens the Copy Case Material Dialog Box (p. 2315).

Delete
deletes the selected materials from the database.

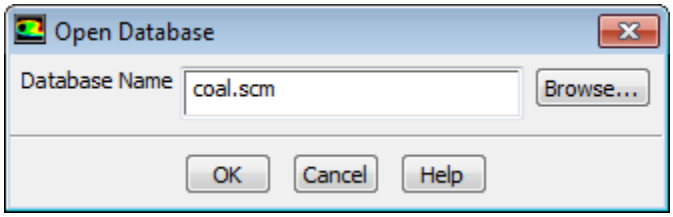
Properties
contains fields for the material properties that are defined for the selected material. These fields are for informational purposes only; they cannot be edited.

When you are viewing the database, not all properties displayed are relevant to your ANSYS Fluent solution. After you copy the material, only properties with relevance to the current physical models will be displayed.

Copy
copies the current material from the global database to the local materials list in the solver.

37.5.3. Open Database Dialog Box

The **Open Database** dialog box is opened by clicking the **User-Defined Database** button in the Create/Edit Materials Dialog Box (p. 2302).



Controls

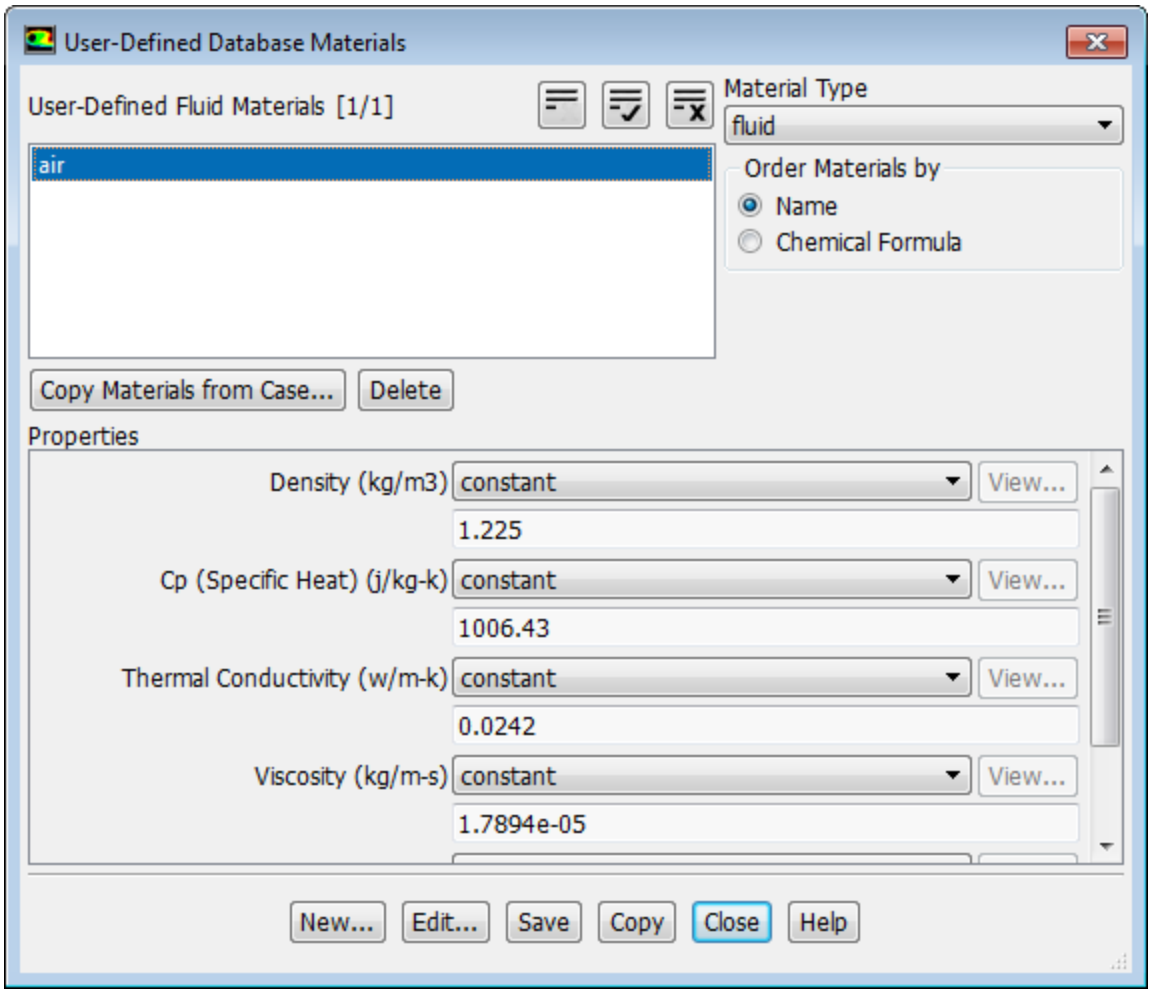
Browse...
opens The Select File Dialog Box (p. 19) where you can select the user-defined database to be used in the current solver session.

Database Name
allows you to enter the path and name of a new database. If you select an existing database, this field displays the path and name of the selected database.

37.5.4. User-Defined Database Materials Dialog Box

The **User-Defined Database Materials** dialog box is opened by clicking **OK** in the Open Database Dialog Box (p. 2313). In this dialog box you can view the user-defined material properties database and copy materials list in the solver. See Viewing Materials in a User-Defined Database (p. 468) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

User-Defined Materials
contains a list of all materials of the selected **Material Type** that are defined in the database. The name of this list will change depending on the selected material type (for example, **User-Defined Fluid Materials**, **User-Defined Solid Materials**, and so on). You can select one or more of these materials to copy to your local list or edit their properties.

Material Type
is a drop-down list containing all of the available material types.

Order Materials By
allows you to order the materials in the **User-Defined Materials** list alphabetically by **Name** or alphabetically by **Chemical Formula**.

Copy Materials from Case...
opens the Copy Case Material Dialog Box (p. 2315).

Delete
deletes the selected materials from the database.

Properties
lists the properties and values of the selected material.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

New...
opens a blank Material Properties Dialog Box (p. 2315) where you can define a new material.

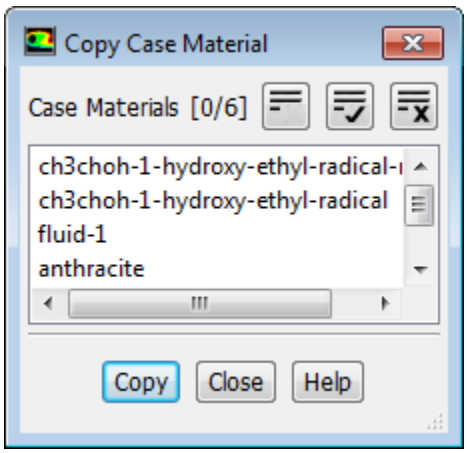
Edit...
opens the Material Properties Dialog Box (p. 2315) displaying the properties of the selected material.

Save
saves the information to the selected database.

Copy
copies the selected material to your local material list. If the material already exists the New Material Name Dialog Box (p. 2317) opens

37.5.5. Copy Case Material Dialog Box

This dialog box is opened by clicking the **Copy Materials from Case...** button in the User-Defined Database Materials Dialog Box (p. 2313).



Controls

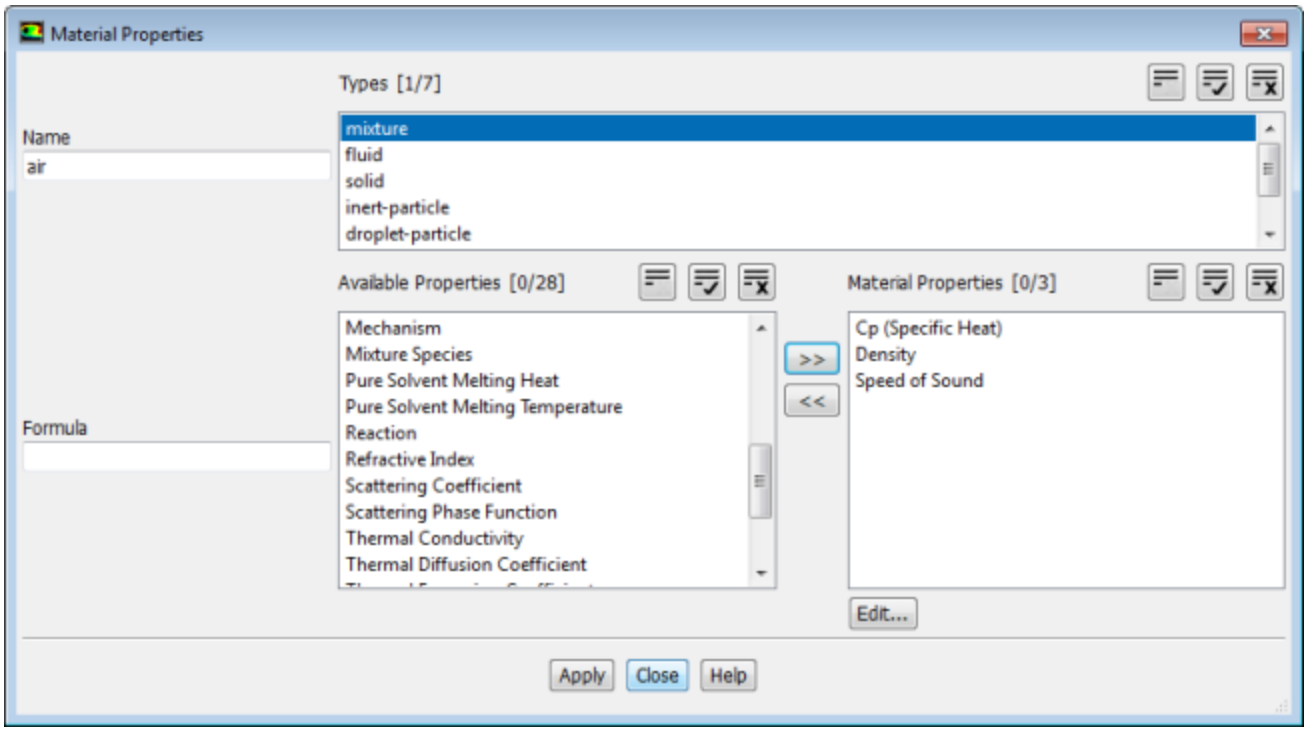
Case Materials
lists all the materials present in your local materials list.

Copy
copies the selected materials to the user-defined database.

37.5.6. Material Properties Dialog Box

This dialog box is opened by clicking **New...** button in the User-Defined Database Materials Dialog Box (p. 2313). See Creating a New Materials Database and Materials (p. 472) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name
specifies the name of the material that you are creating.

Formula
(optional) specifies the chemical formula of the material that you are creating.

Types
allows you to select material type from fluid, solid, inter-particle, droplet-particle, combusting-particle, and mixture materials.

Available Properties
lists properties applicable to the selected material type.

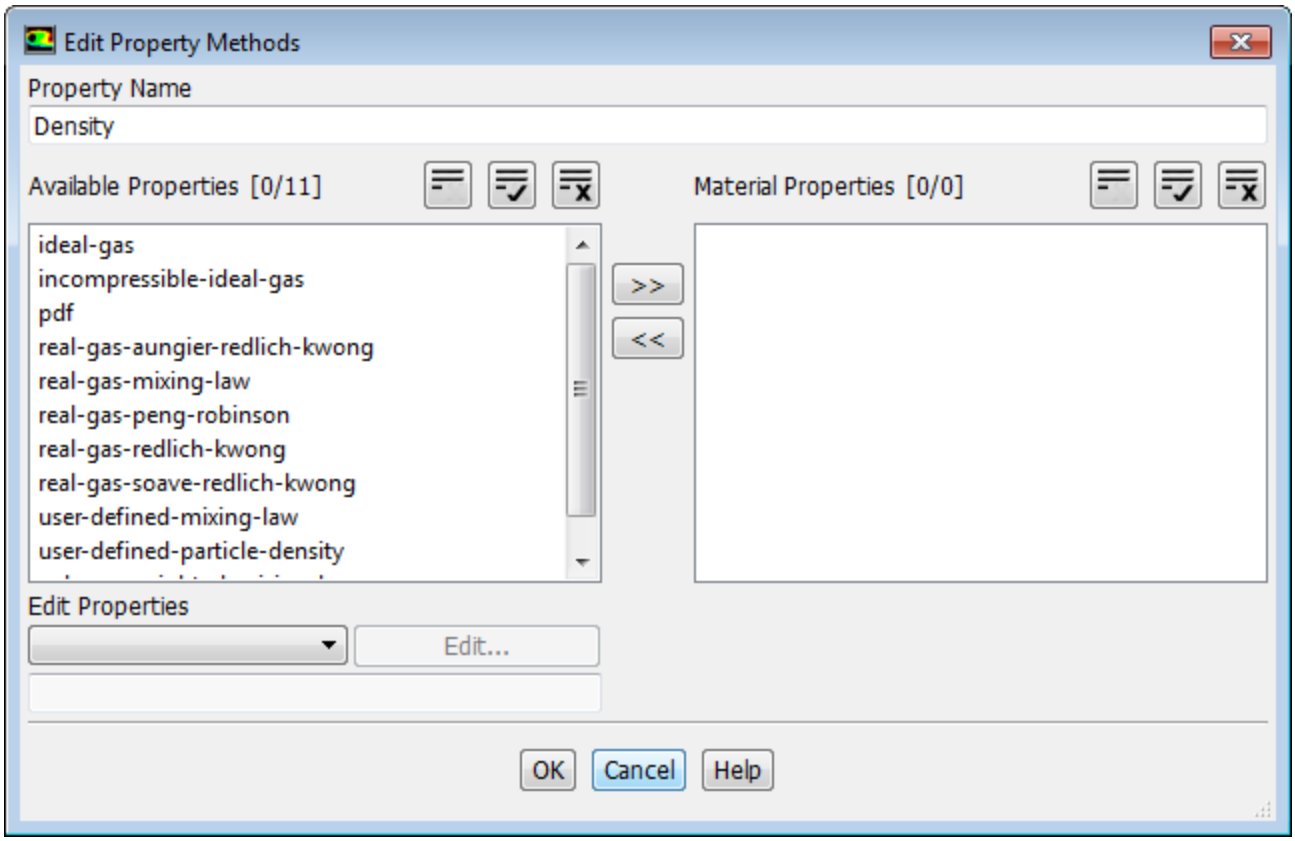
Material Properties
lists properties that you have selected from **Available Properties** list.

Edit...
opens Edit Property Methods Dialog Box (p. 2316) where you can edit the parameters that define a property.

37.5.7. Edit Property Methods Dialog Box

This dialog box is opened by clicking **Edit...** button in the Material Properties Dialog Box (p. 2315). See Creating a New Materials Database and Materials (p. 472) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Property Name
specifies the name of the property that you want to edit.

Available Properties
specifies the methods that can be used to define the selected property.

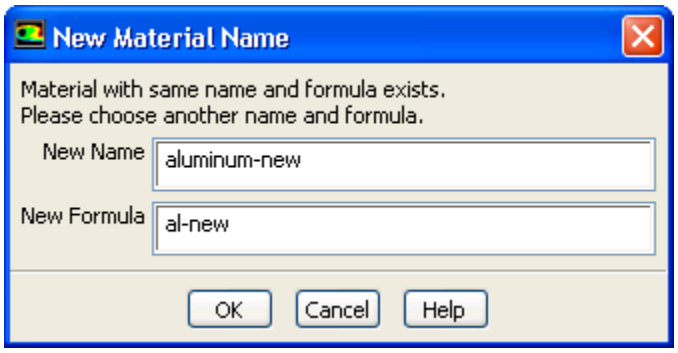
Material Properties
lists properties that you have selected from **Available Properties** list.

Edit Properties
allows you to select the property that you want to edit.

37.5.8. New Material Name Dialog Box

This dialog box is opened by clicking the **Copy** button in the User-Defined Database Materials Dialog Box (p. 2313) when a material is already defined with the same name in the Create/Edit Materials Dialog Box (p. 2302).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



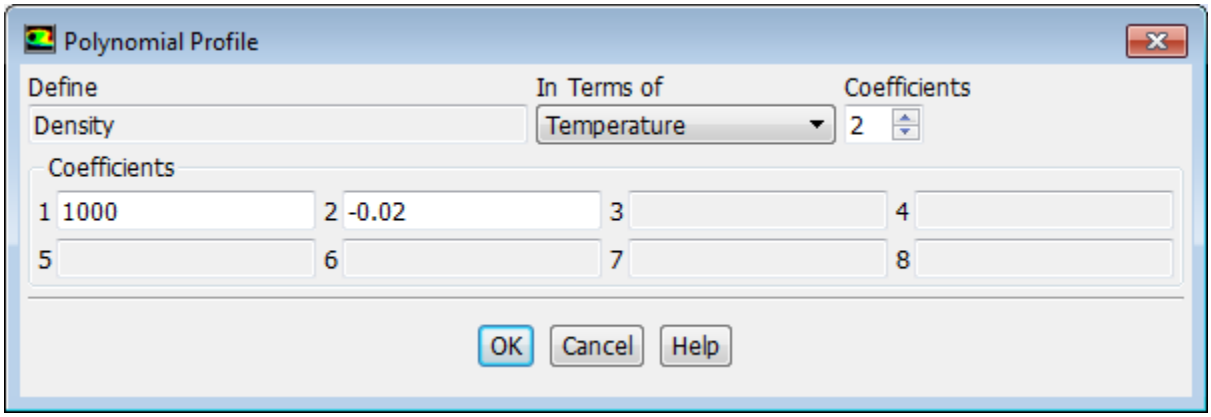
Controls

New Name
allows you to enter the new name for the material you need to copy.

New Formula
allows you to enter the new formula for the material you need to copy.

37.5.9. Polynomial Profile Dialog Box

The **Polynomial Profile** dialog box allows you to define a physical property as a polynomial function of temperature. This dialog box will open when you select **polynomial** in the drop-down list next to a physical property in the Create/Edit Materials Dialog Box (p. 2302). See Inputs for Polynomial Functions (p. 475) for details about the items below.



Controls

Define
shows the property that is being defined as a function of temperature.

In Terms of
shows the independent variable (**Temperature**). The property shown in **Define** will be defined as a polynomial function of temperature.

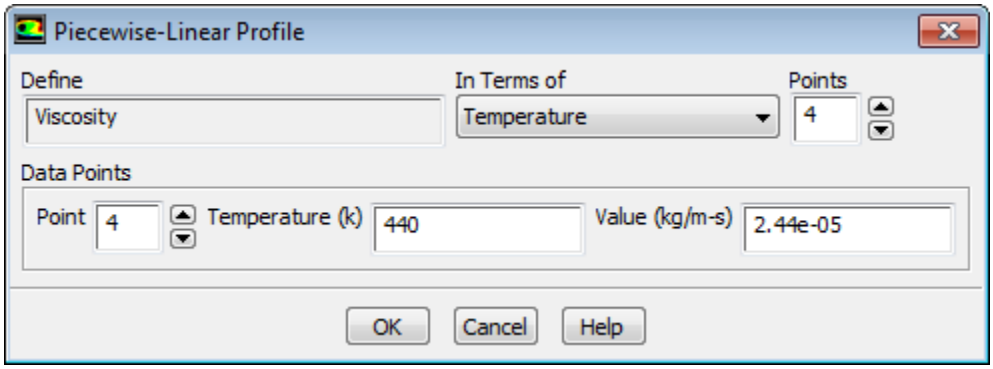
Coefficients
is an integer number entry that indicates the number of coefficients to be defined. You can define up to 8 coefficients.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coefficients
contains real number entries for the number of coefficients set in the **Coefficients** integer number entry above. The number of entries that are editable will be the same as the number of coefficients you requested.

37.5.10. Piecewise-Linear Profile Dialog Box

The **Piecewise-Linear Profile** dialog box allows you to define a physical property as a piecewise-linear function of temperature. This dialog box will open when you select **piecewise-linear** in the drop-down list next to a physical property in the Create/Edit Materials Dialog Box (p. 2302). See Inputs for Piecewise-Linear Functions (p. 476) for details about the items below.



Controls

Define
shows the property that is being defined as a function of temperature.

In Terms of
shows the independent variable (**Temperature**). The property shown in **Define** will be defined as a piecewise-linear function of temperature.

Points
indicates the number of data pairs that will define the piecewise distribution. You can define up to 50 pairs.

Data Points
contains entries for defining the data pairs.

Point
indicates the point for which the data pair (**Temperature**, **Value**) is being defined.

Temperature
is the independent variable.

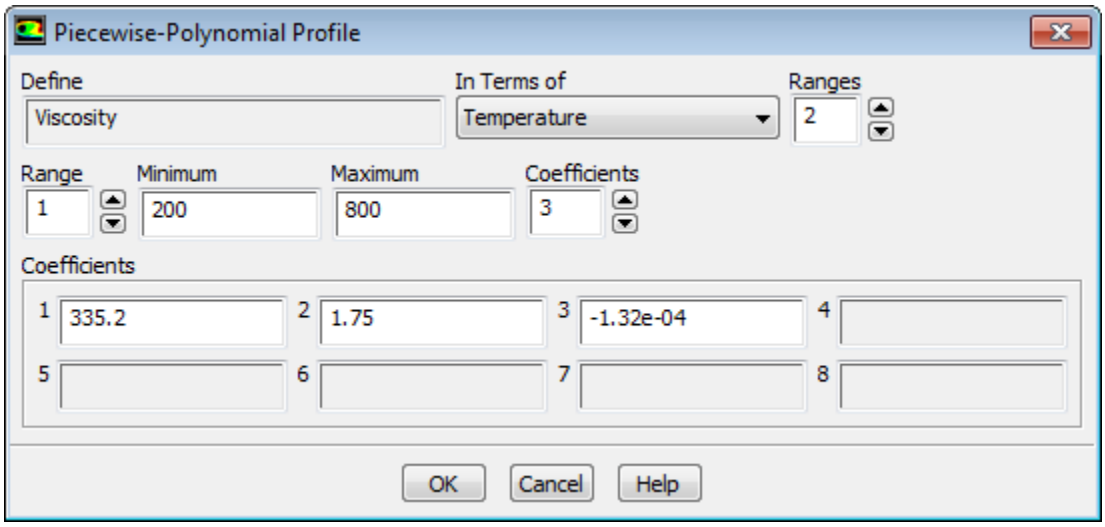
Value
is the dependent variable (the property). In the example dialog box above, **Viscosity** is the variable being defined, as shown in the **Define** field.

37.5.11. Piecewise-Polynomial Profile Dialog Box

The **Piecewise-Polynomial Profile** dialog box allows you to define a physical property as a piecewise-polynomial function of temperature. This dialog box will open when you select **piecewise-polynomial**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in the drop-down list next to a physical property in the Create/Edit Materials Dialog Box (p. 2302). See Inputs for Piecewise-Polynomial Functions (p. 478) for details about the items below.



Controls

Define
shows the property that is being defined as a function of temperature.

In Terms of
shows the independent variable (**Temperature**). The property shown in **Define** will be defined as a polynomial function of temperature.

Ranges
sets the number of temperature ranges for which you will define polynomial functions. You can define up to 3 ranges.

Range
indicates the temperature range for which you are defining the polynomial function.

Minimum, Maximum
set the minimum and maximum temperatures for the specified **Range**.

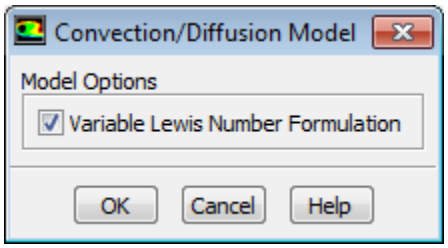
Coefficients
is an integer number entry that indicates the number of coefficients to be defined for the specified **Range**. You can define up to 8 coefficients.

Coefficients
contains real number entries for the number of coefficients set in the **Coefficients** integer number entry above. The number of entries that are editable will be the same as the number of coefficients you requested for the specified **Range**.

37.5.12. Convection/Diffusion Model Dialog Box

The **Convection/Diffusion Model** dialog box allows you to enable the calculation of the Spalding heat number when the **convection/diffusion-controlled** option is selected for **Vaporization Model** in the Create/Edit Materials Dialog Box (p. 2302). This dialog box opens when you click **Edit...** next to **Vaporization Model**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

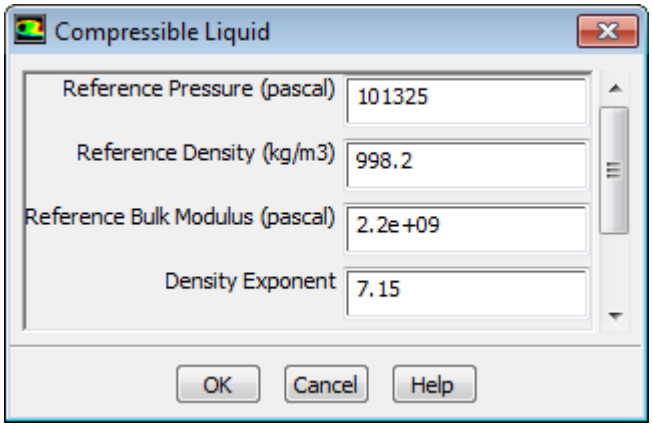


Model Options

allows you to enable **Variable Lewis Number Formulation** if you want to compute the Spalding heat number from Equation 16.115 in the *Fluent Theory Guide*.

37.5.13. Compressible Liquid Dialog Box

The **Compressible Liquid** dialog box allows you to model liquid compressibility for high pressure applications using the Tait equation of state. This dialog box will open when you select **compressible-liquid** in the drop-down list next to one of the **Properties** in the Create/Edit Materials Dialog Box (p. 2302). See Compressible Liquid Density Method (p. 480) for details about using the items below.



Controls

Reference Pressure

sets the reference pressure.

Reference Density

sets the reference density.

Reference Bulk Modulus

sets the reference bulk modulus.

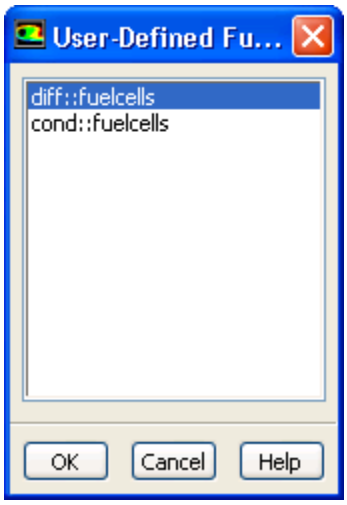
Density Exponent

sets the density exponent.

37.5.14. User-Defined Functions Dialog Box

The **User-Defined Functions** dialog box allows you to choose which user-defined function is to be used to define a material property. This dialog box will open when you select **user-defined** in the drop-down list next to one of the **Properties** in the Create/Edit Materials Dialog Box (p. 2302). See the separate Fluent Customization Manual for details about user-defined functions.

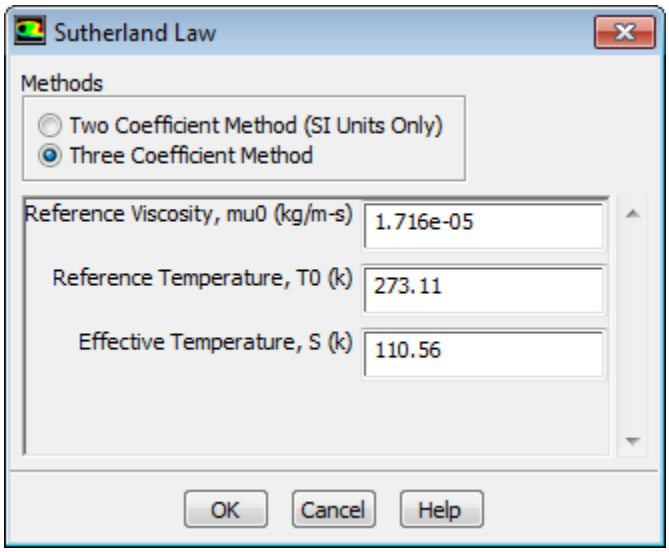
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The list will contain all available user-defined functions.

37.5.15. Sutherland Law Dialog Box

The **Sutherland Law** dialog box allows you to set the coefficients for Sutherland's law for viscosity. This dialog box will open when you select **sutherland** in the drop-down list next to **Viscosity** in the Create/Edit Materials Dialog Box (p. 2302). See Sutherland Viscosity Law (p. 489) for details about the items below.



Controls

Methods
contains options for selecting the **Two Coefficient Method** or the **Three Coefficient Method**.

C1, C2
set the coefficients C_1 and C_2 in Equation 7.23 (p. 489) in SI units. These inputs will appear if you select the **Two Coefficient Method**.

Reference Viscosity
sets the reference viscosity μ_0 in Equation 7.24 (p. 489). This input will appear if you select the **Three Coefficient Method**.

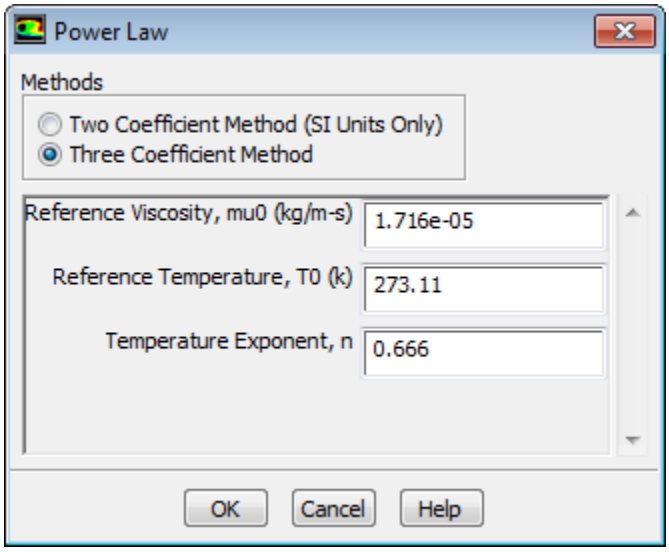
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reference Temperature
sets the reference temperature T_0 in Equation 7.24 (p. 489). This input will appear if you select the **Three Coefficient Method**.

Effective Temperature
sets the effective temperature S in Equation 7.24 (p. 489). This input will appear if you select the **Three Coefficient Method**.

37.5.16. Power Law Dialog Box

The **Power Law** dialog box allows you to set the coefficients for the power law for viscosity. This dialog box will open when you select **power-law** in the drop-down list next to **Viscosity** in the Create/Edit Materials Dialog Box (p. 2302). See Power-Law Viscosity Law (p. 489) for details about the items below.



Controls

Methods
contains options for selecting the **Two Coefficient Method** or the **Three Coefficient Method**.

B
sets the coefficient B in Equation 7.25 (p. 490) in SI units. This input will appear if you select the **Two Coefficient Method**.

Reference Viscosity
sets the reference viscosity μ_0 in Equation 7.26 (p. 490). This input will appear if you select the **Three Coefficient Method**.

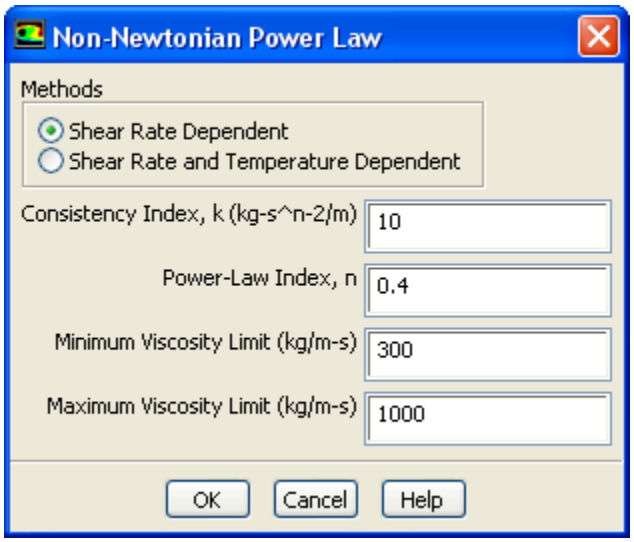
Reference Temperature
sets the reference temperature T_0 in Equation 7.26 (p. 490). This input will appear if you select the **Three Coefficient Method**.

Temperature Exponent
sets the temperature exponent n in Equation 7.25 (p. 490) or Equation 7.26 (p. 490), depending on your **Method** selection. If you are using the **Two Coefficient Method**, this input must be in SI units.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.5.17. Non-Newtonian Power Law Dialog Box

The **Non-Newtonian Power Law** dialog box allows you to set the parameters for the non-Newtonian power law for viscosity. This dialog box will open when you select **non-newtonian-power-law** in the drop-down list next to **Viscosity** in the Create/Edit Materials Dialog Box (p. 2302). See Power Law for Non-Newtonian Viscosity (p. 493) for details about the items below.



Controls

Methods
allows you to select the type of dependency on the viscosity.

Shear Rate Dependent
is where the viscosity is dependent on the shear rate.

Shear Rate and Temperature Dependent
is where the viscosity is dependent on the shear rate and the temperature.

Consistency Index
sets the consistency index k in Equation 7.38 (p. 493).

Power-Law Index
sets the power-law index n in Equation 7.38 (p. 493).

Minimum Viscosity Limit, Maximum Viscosity Limit
set the minimum and maximum viscosity limits.

Reference Temperature
sets the reference temperature.

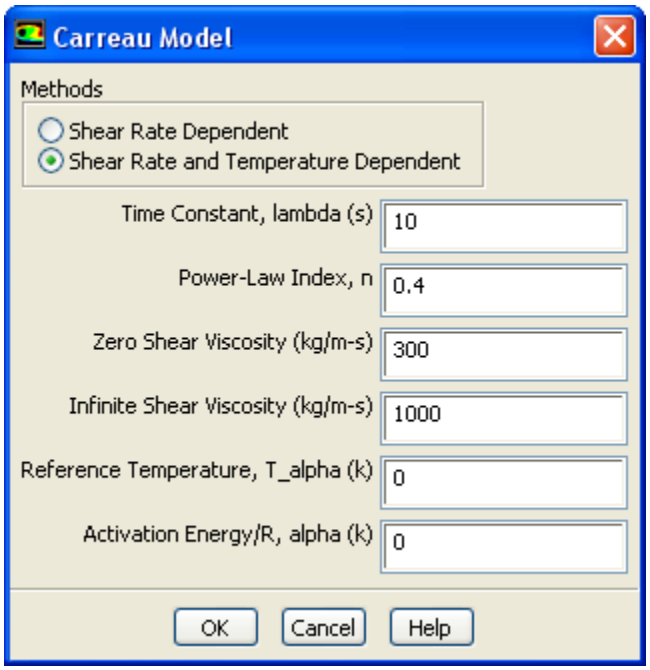
Activation Energy/R, alpha
is the ratio of the activation energy to the thermodynamic constant α in Equation 7.37 (p. 493).

37.5.18. Carreau Model Dialog Box

The **Carreau Model** dialog box allows you to set the parameters for the non-Newtonian Carreau model for viscosity. This dialog box will open when you select **carreau** in the drop-down list next to **Viscosity**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in the Create/Edit Materials Dialog Box (p. 2302). See The Carreau Model for Pseudo-Plastics (p. 494) for details about the items below.



Controls

Methods
allows you to select the type of dependency on the viscosity.

Shear Rate Dependent
is where the viscosity is dependent on the shear rate.

Shear Rate and Temperature Dependent
is where the viscosity is dependent on the shear rate and the temperature.

Time Constant, lambda
sets the time constant λ in Equation 7.39 (p. 494).

Power-Law Index
sets the power-law index n in Equation 7.39 (p. 494).

Zero Shear Viscosity, Infinite Shear Viscosity
set the zero and infinite shear viscosity limits η_0 and η_∞ in Equation 7.39 (p. 494).

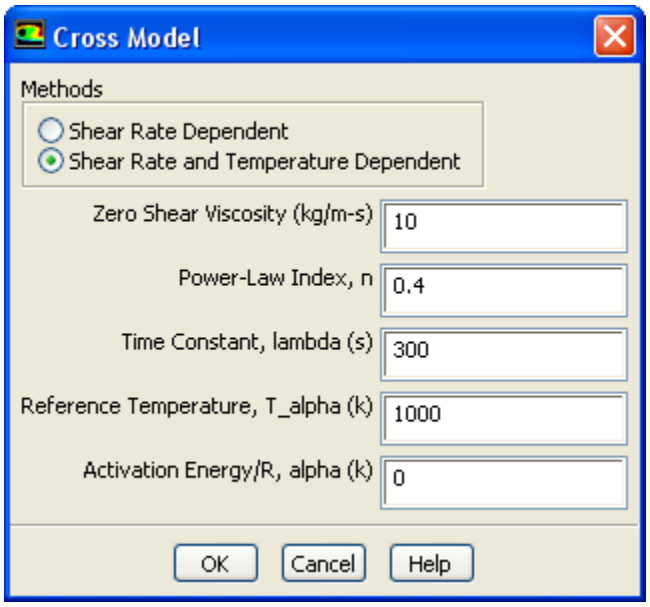
Reference Temperature, T_alpha
sets the reference temperature T_0 in Equation 7.39 (p. 494).

Activation Energy/R, alpha
is the ratio of the activation energy to the thermodynamic constant α in Equation 7.37 (p. 493).

37.5.19. Cross Model Dialog Box

The **Cross Model** dialog box allows you to set the parameters for the non-Newtonian Cross model for viscosity. This dialog box will open when you select **cross** in the drop-down list next to **Viscosity** in the Create/Edit Materials Dialog Box (p. 2302). See Cross Model (p. 495) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Methods
allows you to select the type of dependency on the viscosity.

Shear Rate Dependent
is where the viscosity is dependent on the shear rate.

Shear Rate and Temperature Dependent
is where the viscosity is dependent on the shear rate and the temperature.

Zero Shear Viscosity
sets the zero shear viscosity limit η_0 in Equation 7.40 (p. 495).

Power-Law Index
sets the power-law index n in Equation 7.40 (p. 495).

Time Constant
sets the time constant λ in Equation 7.40 (p. 495).

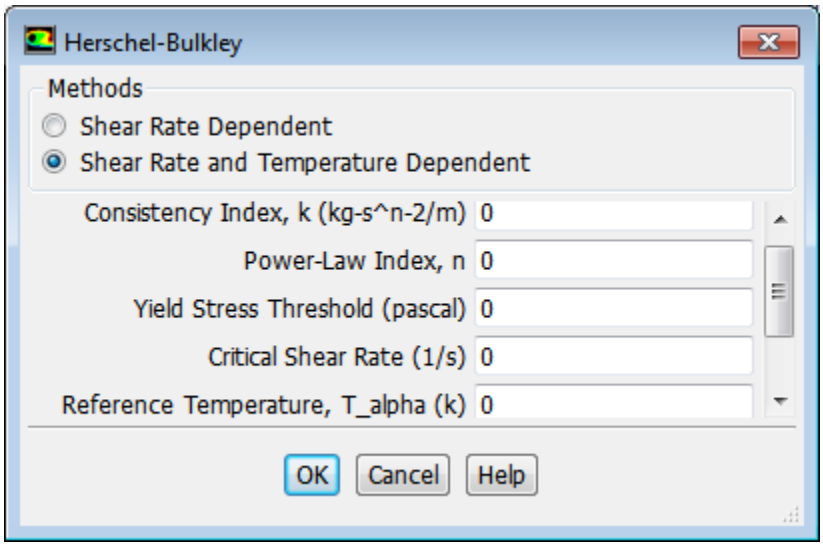
Reference Temperature, T_alpha
sets the reference temperature T_0 in Equation 7.39 (p. 494).

Activation Energy/R, alpha
is the ratio of the activation energy to the thermodynamic constant α in Equation 7.37 (p. 493).

37.5.20. Herschel-Bulkley Dialog Box

The **Herschel-Bulkley** dialog box allows you to set the parameters for the non-Newtonian Herschel-Bulkley model for viscosity. This dialog box will open when you select **herschel-bulkley** in the drop-down list next to **Viscosity** in the Create/Edit Materials Dialog Box (p. 2302). See Herschel-Bulkley Model for Bingham Plastics (p. 495) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Methods
allows you to select the type of dependency on the viscosity.

Shear Rate Dependent
is where the viscosity is dependent on the shear rate.

Shear Rate and Temperature Dependent
is where the viscosity is dependent on the shear rate and the temperature.

Consistency Index
sets the consistency index k in Equation 7.42 (p. 496).

Power-Law Index
sets the power-law index n in Equation 7.42 (p. 496).

Yield Stress Threshold
sets the yield stress threshold τ_0 in Equation 7.42 (p. 496).

Critical Shear Rate
set the critical shear rate $\dot{\gamma}_c$ in Equation 7.42 (p. 496).

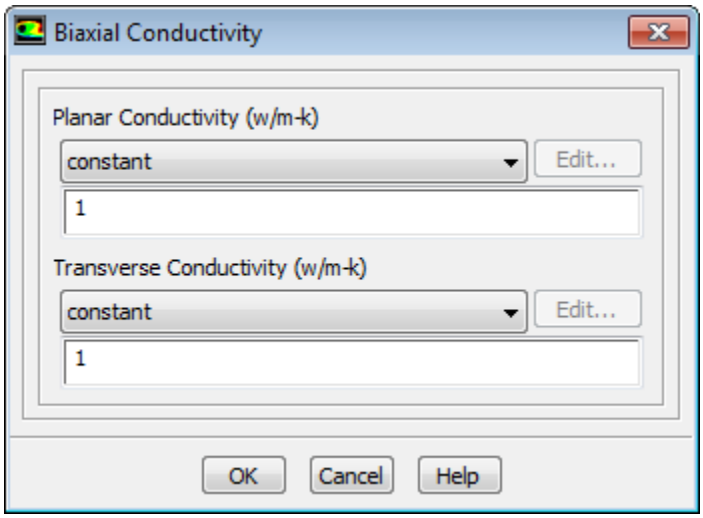
Reference Temperature, T_alpha
sets the reference temperature T_α in Equation 7.37 (p. 493).

Activation Energy/R, alpha
is the ratio of the activation energy to the thermodynamic constant α in Equation 7.37 (p. 493).

37.5.21. Biaxial Conductivity Dialog Box

The **Biaxial Conductivity** dialog box allows you to define a biaxial orthotropic thermal conductivity, which is applicable to solid materials used for the wall shell conduction model. This dialog box will open when you select **biaxial** in the drop-down list next to **Thermal Conductivity** in the Create/Edit Materials Dialog Box (p. 2302). See Biaxial Thermal Conductivity (p. 502) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

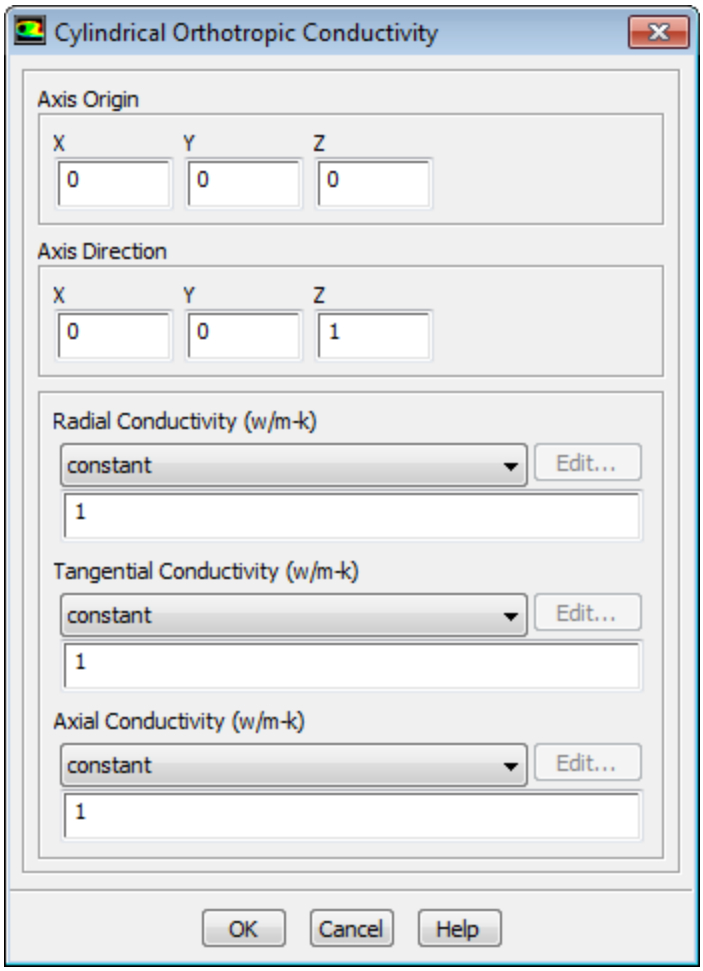
Planar Conductivity
specifies the conductivity within the shell (or solid) region.

Transverse Conductivity
specifies the conductivity normal to the surface of the solid region.

37.5.22. Cylindrical Orthotropic Conductivity Dialog Box

The **Cylindrical Orthotropic Conductivity** dialog box allows you to define an orthotropic thermal conductivity in cylindrical coordinates. This dialog box will open when you select **cyl-orthotropic** in the drop-down list next to **Thermal Conductivity** in the Create/Edit Materials Dialog Box (p. 2302). See Cylindrical Orthotropic Thermal Conductivity (p. 504) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Axis Origin

allows you to specify the origin of the cylindrical coordinate system.

X, Y, Z

specify the X, Y, and (for 3D cases) Z coordinates.

Axis Direction

(3D only) allows you to specify the direction of the axis.

X, Y, Z

specify 1 against the direction of the axis.

Radial Conductivity

specifies the conductivity in the radial direction.

Tangential Conductivity

specifies the conductivity in the tangential direction.

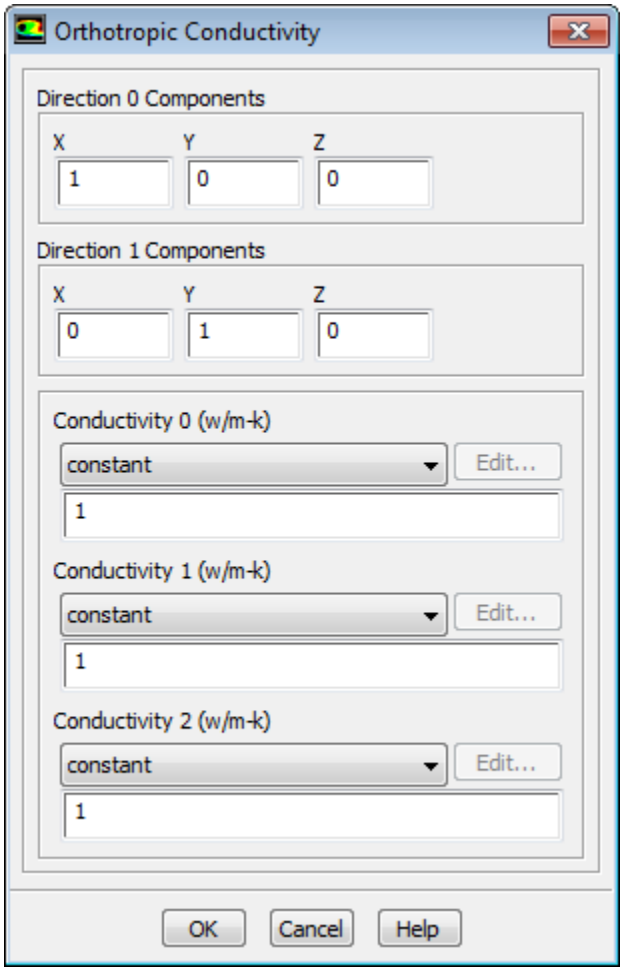
Axial Conductivity

(3D only) specifies the conductivity in the axial direction.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.5.23. Orthotropic Conductivity Dialog Box

The **Orthotropic Conductivity** dialog box allows you to define an orthotropic thermal conductivity for a solid material, using principal directions that do not have to aligned with the global coordinate system of the simulation. This dialog box will open when you select **orthotropic** in the drop-down list next to **Thermal Conductivity** in the Create/Edit Materials Dialog Box (p. 2302). See Orthotropic Thermal Conductivity (p. 503) for details about the items below.



Controls

Direction 0 Components, Direction 1 Components

specify the directions $\hat{e}_\xi$ and $\hat{e}_\eta$ in Equation 7.53 (p. 503) as **X,Y,Z** vectors. For 2D cases, only **Direction 0 Components** will appear.

Conductivity 0, Conductivity 1, Conductivity 2

specify $\left(k_\xi\right)$, $\left(k_\eta\right)$, and $\left(k_\zeta\right)$ in Equation 7.53 (p. 503) as **constant**, **polynomial**, **piecewise-linear**, or **piecewise-polynomial** functions of temperature. For 2D cases, only **Conductivity 0** and **Conductivity 1** will appear.

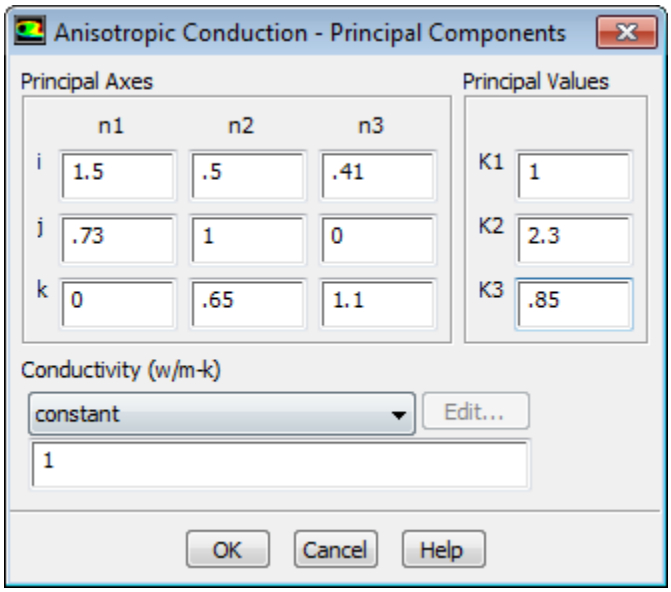
Edit...

opens the appropriate dialog box so you can enter a temperature-dependent conductivity. (This button will be unavailable if you specify a **constant** conductivity.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.5.24. Anisotropic Conductivity - Principal Components Dialog Box

The **Anisotropic Conductivity - Principal Components** dialog box allows you to define an anisotropic thermal conductivity using the components of the principal axes (which do not have to be orthogonal and/or aligned with the global coordinate system) and the associated principal values. This dialog box will open when you select **principal-axes-values** in the drop-down list next to **Thermal Conductivity** in the Create/Edit Materials Dialog Box (p. 2302). See Principal Axes and Principal Values (p. 506) for details about the items that follow.



Controls

Principal Axes

specifies the $\hat{i}$, $\hat{j}$, and $\hat{k}$ components of the $\vec{n}^1$, $\vec{n}^2$, and $\vec{n}^3$ vectors (in Equation 7.54 (p. 506)) that define the principal axes of the anisotropic material. Note that these vectors must be linearly independent.

Principal Values

specifies the principal values (K_1, K_2, and K_3 in Equation 7.57 (p. 506)) associated with the principal axes.

Conductivity

specifies the value of k (in Equation 7.57 (p. 506)) as a **constant**, function of temperature (**polynomial**, **piecewise-linear**, **piecewise-polynomial**), or **user-defined** function.

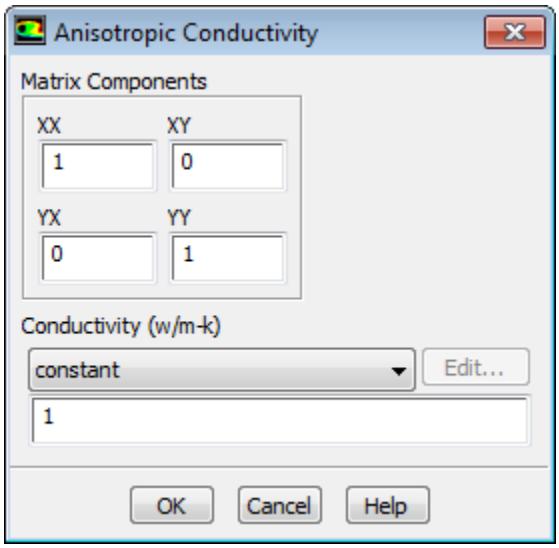
Edit...

opens the appropriate dialog box for the definition of a temperature-dependent or user-defined conductivity. (This button will be unavailable if you specify a **constant** conductivity.)

37.5.25. Anisotropic Conductivity Dialog Box

The **Anisotropic Conductivity** dialog box allows you to define a general anisotropic thermal conductivity. This dialog box will open when you select **anisotropic** in the drop-down list next to **Thermal Conductivity** in the Create/Edit Materials Dialog Box (p. 2302). See Anisotropic Thermal Conductivity (p. 501) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Matrix Components
specify the components of the matrix $\hat{e}_{ij}$ in Equation 7.52 (p. 501).

Conductivity
specifies the value of k in Equation 7.52 (p. 501) as a **constant**, function of temperature (**polynomial**, **piecewise-linear**, **piecewise-polynomial**), or **user-defined** function.

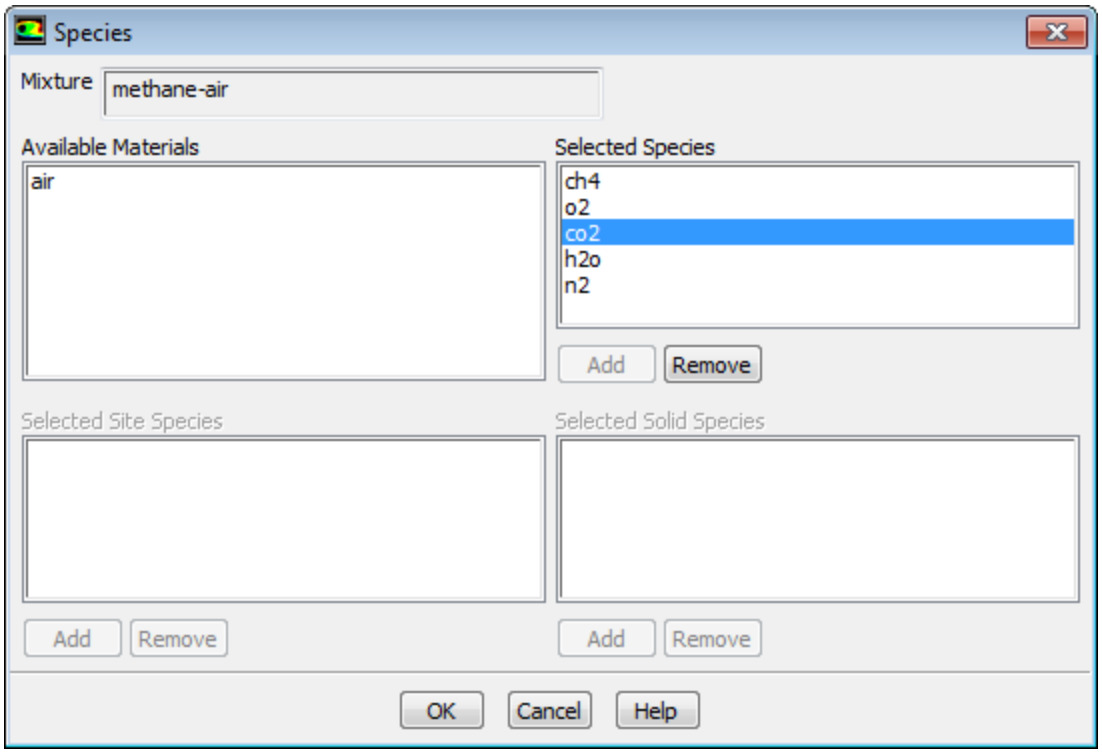
Edit...
opens the appropriate dialog box so you can enter a temperature-dependent or user-defined conductivity. (This button will be unavailable if you specify a **constant** conductivity.)

37.5.26. Species Dialog Box

The **Species** dialog box (opened by clicking the **Edit...** button next to **Mixture Species** in the Create/Edit Materials Dialog Box (p. 2302))) allows you to define the species that comprise a mixture material. See Defining the Species in the Mixture (p. 973) for details about the items below.

(Note that the **Species** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Mixture

shows the name of the mixture material for which you are defining the species. This field is not editable.

Available Materials

is a list of all of the materials to be a component in the mixture material by selecting it and clicking the **Add** button below the **Selected Species** or **Selected Surface Species** list. To add a material to the **Available Materials** list, use the Fluent Database Materials Dialog Box (p. 2312) to copy the fluid material to local storage.

Selected Species

is a list of all the fluid-phase species in the mixture. To add a material to the list, select it in the **Available Materials** list and click the **Add** button below the **Selected Species** list. To remove a material, select it in the **Selected Species** list and click **Remove**. See Overview of the Species Dialog Box (p. 974) for more information.

Selected Solid Species

is a list of all the solid species in the mixture. To add a material to the list, select it in the **Available Materials** list and click the **Add** button below the **Selected Solid Species** list. To remove a material, select it in the **Selected Solid Species** list and click **Remove**. See Overview of the Species Dialog Box (p. 974) for more information.

Selected Site Species

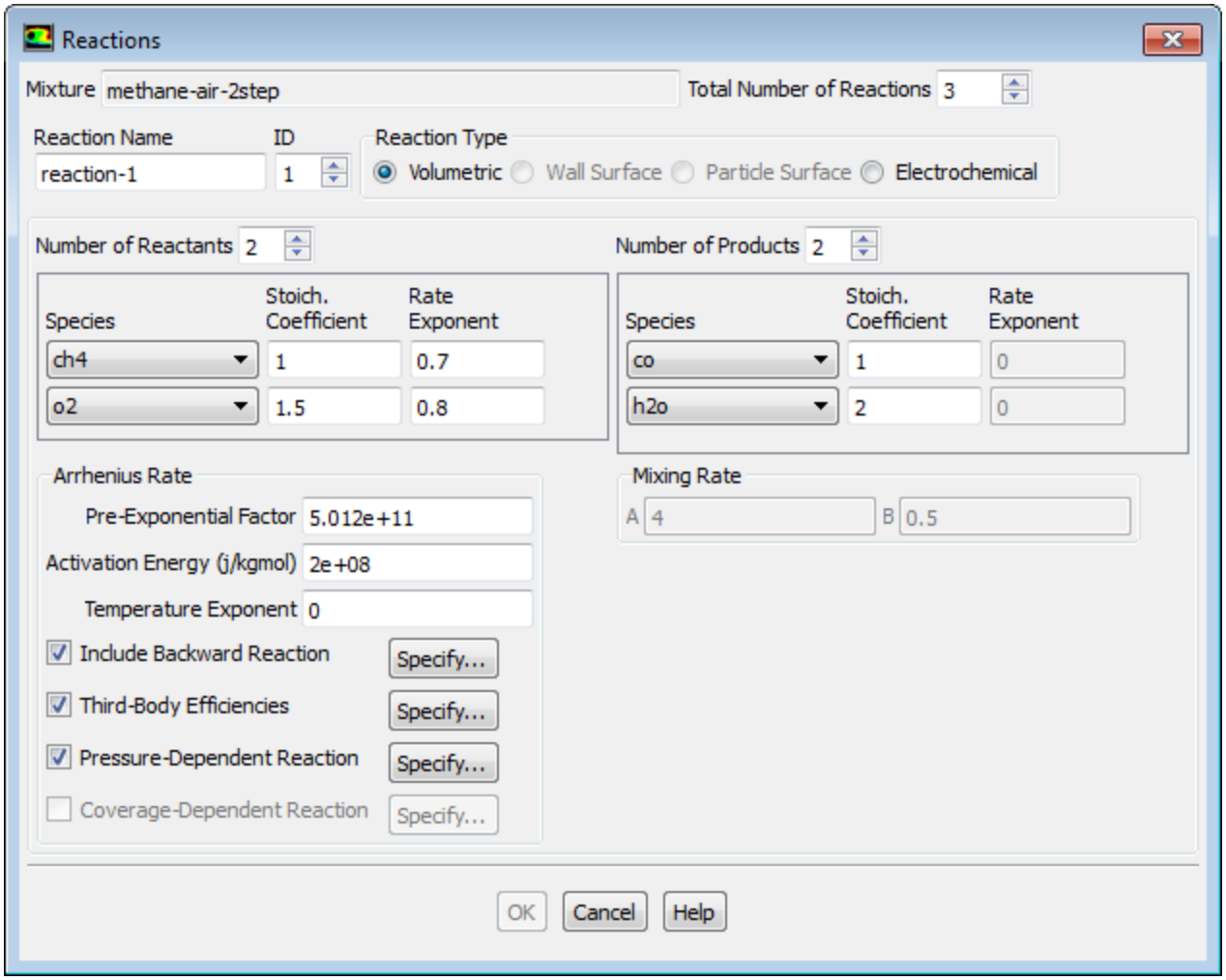
is a list of all the site species in the mixture. To add a material to the list, select it in the **Available Materials** list and click the **Add** button below the **Selected Site Species** list. To remove a material, select it in the **Selected Site Species** list and click **Remove**. See Overview of the Species Dialog Box (p. 974) for more information.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.5.27. Reactions Dialog Box

The **Reactions** dialog box (opened by clicking the **Edit...** button next to **Reaction** in the Create/Edit Materials Dialog Box (p. 2302)) allows you to define the reactions that comprise a mixture material. See Defining Reactions (p. 977) for details about using this dialog box.

(Note that the **Reactions** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.)



Controls

Mixture

shows the name of the mixture material for which you are defining the species. This field is not editable.

Total Number of Reactions

sets the total number of reactions (fluid-phase reactions and surface reactions occurring at wall boundaries). Use the arrows to change the value, or type in the value and press RETURN.

Reaction Name

contains the name of the reaction.

ID

sets the number of the reaction you want to define. (Again, if you type in the value be sure to press RETURN.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reaction Type
contains options that allow you to specify the type of reaction.

Volumetric
specifies, if enabled, that the current reaction is a volumetric reaction.

Wall Surface
specifies, if enabled, that the current reaction is a wall surface reaction.

Particle Surface
specifies, if enabled, that the current reaction is a particle surface reaction.

Number of Reactants
indicates the number of reactants in the specified reaction.

Species
contains drop-down lists of all species in the mixture. (The number of lists will be equal to the **Number of Reactants**.) Select each reactant in one of these lists.

Stoich. Coefficient
specifies the stoichiometric coefficient of the reactant species in the reaction.

Rate Exponent
specifies the rate constant for the reactant species in the reaction.

Number of Products
indicates the number of products in the specified reaction.

Species
contains drop-down lists of all species in the mixture. (The number of lists will be equal to the **Number of Products**.) Select each product in one of these lists.

Stoich. Coefficient
specifies the stoichiometric coefficient of the product species in the reaction.

Rate Exponent
specifies the rate constant for the product species in the reaction.

Arrhenius Rate
contains inputs related to the Arrhenius rate. (If you have chosen **Eddy-Dissipation** for the **Turbulence-Chemistry Interaction** in the Species Model Dialog Box (p. 2216), these inputs are not required.)

Pre-exponential Factor
is the constant A_r in Equation 7.9 in the Theory Guide. The units of A_r depend on the other rate constant inputs, but must be defined such that the units of the reaction rate $R_{i,r}$ (Equation 7.5 in the Theory Guide) are in (kg/m^3-s) if you are using SI units.

Important

It is important to note that if you have selected the British units system, the Arrhenius factor should still be specified in SI units. This is because ANSYS Fluent applies no conversion factor to the value you enter for A_r (the conversion factor is 1.0) when

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

you work in British units, as the correct conversion factor depends on your values for $\nu_{j,r}$, β_r, and so on.

Activation Energy
is the constant E_r in the forward rate constant expression, Equation 7.9 in the Theory Guide).

Temperature Exponent
is the value for the constant β_r in Equation 7.9 in the Theory Guide.

Include Backward Reaction
specifies that the reaction is reversible. By default, the backward reaction rate constant will be computed from Equation 7.9 in the *Fluent Theory Guide*. You can specify your own backward reaction rate parameters in the Backward Reaction Parameters Dialog Box (p. 2337). In this case, the backward reaction rate constant will be computed from Equation 7.14 in the *Fluent Theory Guide*.

Third-Body Efficiencies
allows you to specify and use third-body efficiencies ($\gamma_{j,r}$ in Equation 7.8 in the Theory Guide). These inputs are optional. (This item is available only if you have selected **Volumetric** for the **Reaction Type**.)

Pressure-Dependent Reaction
enables the modeling of a pressure fall-off reaction. See Inputs for Reaction Definition (p. 977) **Laminar Finite-Rate** or **Eddy-Dissipation Concept** for the **Turbulence-Chemistry Interaction** in the Species Model Dialog Box (p. 2216) and have selected **Volumetric** for the **Reaction Type**.)

Coverage-Dependent Reaction
is used when modeling **Wall Surface** reactions with site-balancing and reaction rates depend on site coverages.

Specify...
opens the Backward Reaction Parameters Dialog Box (p. 2337), Third-Body Efficiency Dialog Box (p. 2338), the Pressure-Dependent Reaction Dialog Box (p. 2339), or Coverage-Dependent Reaction Dialog Box (p. 2341) in which you can specify the backward rate parameters for the reversible reaction, third-body efficiencies, pressure-dependent reaction parameters, or coverage parameters.

Mixing Rate
contains inputs related to the mixing rate. (If you have chosen **Laminar Finite-Rate** or **Eddy-Dissipation Concept** for the **Turbulence-Chemistry Interaction** in the Species Model Dialog Box (p. 2216), these inputs are not required.)

A
is the constant A in the turbulent mixing rate (Equation 7.25 and Equation 7.26 in the Theory Guide) when it is applied to a species that appears as a reactant in this reaction. The default setting of 4.0 is based on the empirically derived values given by Magnussen and Hjertager [64] (p. 2890).

B
is the constant B in the turbulent mixing rate (Equation 7.26 in the Theory Guide) when it is applied to a species that appears as a product in this reaction. The default setting of 0.5 is based on the empirically derived values given by Magnussen and Hjertager [64] (p. 2890).

Particle Surface Reaction
contains inputs related to a particle surface reaction. See User Inputs for Particle Surface Reactions (p. 1001) for details. (This section will appear only if you have selected **Particle Surface** for the **Reaction Type**.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Diffusion Limited Species
is a drop-down list that allows you to select the species for which the concentration gradient between the bulk and the particle surface is the largest when there is more than one gaseous reactant taking part in the particle surface reaction. See User Inputs for Particle Surface Reactions (p. 1001) for details.

Diffusion Rate Constant
is the constant $C_{1,r}$ in Equation 7.71 in the Theory Guide.

Effectiveness Factor
is the constant η_r in Equation 7.69 in the Theory Guide.

Butler-Volmer/Tafel Parameters
contains inputs related to an electrochemical reaction. See User Inputs for Electrochemical Reactions (p. 1004) for details. (This group box appears only if **Electrochemical** is selected for **Reaction Type**.)

Tafel Parameters
indicates whether the Bulter-Volmer equation (Equation 7.80 in the *Fluent Theory Guide*) or the Tafel equation (Equation 7.81 in the *Fluent Theory Guide*) is used in the kinetics calculation.

Anodic Transfer Coefficient
is the value of α_a used in Equation 7.80 in the *Fluent Theory Guide*. This item appears if the **Tafel Parameters** check box is cleared.

Cathodic Transfer Coefficient
is the value of α_c used in Equation 7.80 in the *Fluent Theory Guide*. This item appears if the **Tafel Parameters** check box is cleared.

Anodic Tafel Slope
is the value of $\beta_{a,r}$ in Equation 7.81 in the *Fluent Theory Guide*. This item appears only if the **Tafel Parameters** check box is selected.

Cathodic Tafel Slope
is the value of $\beta_{c,r}$ in Equation 7.81 in the *Fluent Theory Guide*. This item appears only if the **Tafel Parameters** check box is selected.

Exchange Current Density
is the value of i_o used in Equation 7.80 in the *Fluent Theory Guide*

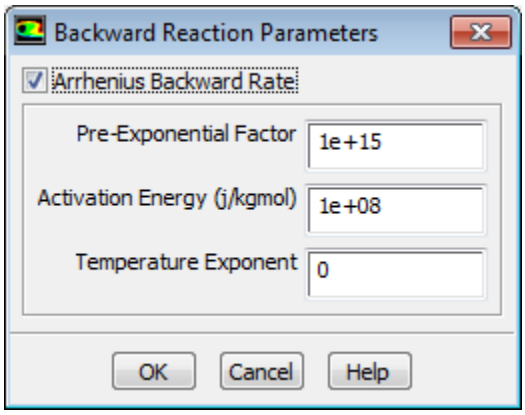
Equilibrium Potential
is the value of E_{eq} used in Equation 7.82 in the *Fluent Theory Guide*

Specify Reference Mass Fractions...
opens the Reference Mass Fractions Dialog Box (p. 2341).

37.5.28. Backward Reaction Parameters Dialog Box

The **Backward Reaction Parameters** dialog box (opened by clicking **Specify...** next to **Include Backward Reaction** in the Reactions Dialog Box (p. 2334)) allows you to specify your custom backward rate parameters for a reversible reaction. See Inputs for Reaction Definition (p. 977) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Arrhenius Backward Rate
allows you to specify backward rate parameters.

Pre-Exponential Factor
is the constant $A_{b,r}$ in Equation 7.14 in the *Fluent Theory Guide*.

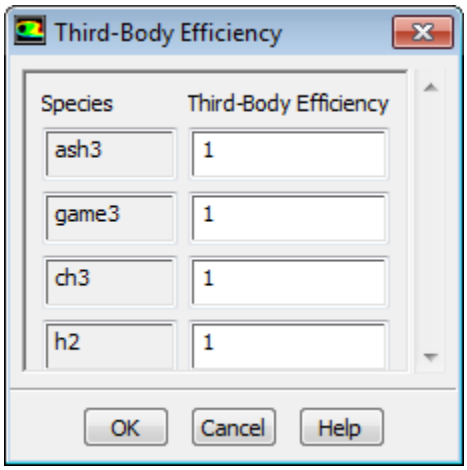
Activation Energy
is the constant $E_{b,r}$ in the backward rate constant expression in Equation 7.14 in the *Fluent Theory Guide*.

Temperature Exponent
is the constant $\beta_{b,r}$ in Equation 7.14 in the *Fluent Theory Guide*.

37.5.29. Third-Body Efficiency Dialog Box

The **Third-Body Efficiency** dialog box (opened by clicking **Specify...** next to the **Third-Body Efficiencies** button in the Reactions Dialog Box (p. 2334)) allows you to specify the third-body efficiencies for each species in the mixture, to be used in Equation 7.8 in the Theory Guide. See Defining Reactions (p. 977) for details.

(Note that the **Third-Body Efficiency** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.)



Controls

Species
displays the name of each species in the mixture.

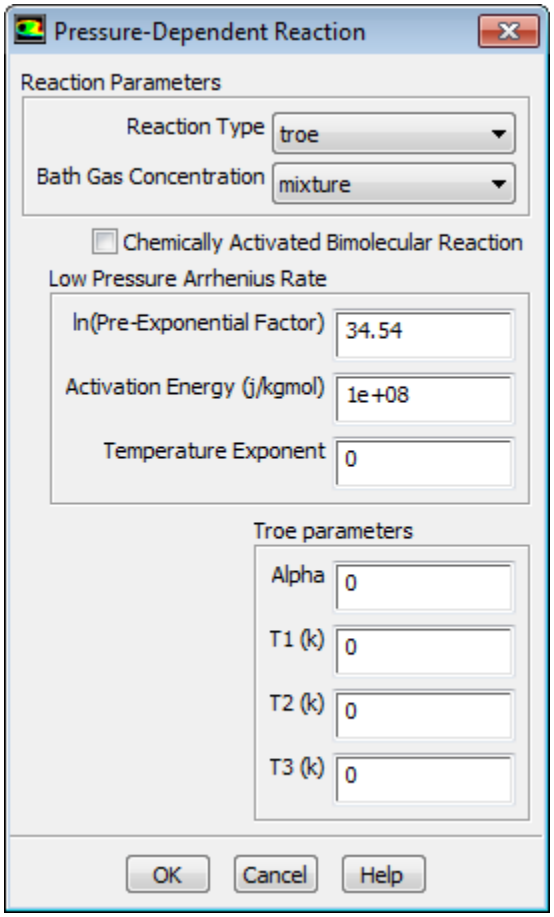
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Third-body Efficiency
specifies the third-body efficiency for each species.

37.5.30. Pressure-Dependent Reaction Dialog Box

The **Pressure-Dependent Reaction** dialog box (opened by clicking **Specify...** under **Pressure-Dependent Reaction** in the Reactions Dialog Box (p. 2334)) allows you to specify parameters for a pressure fall-off reaction. See Inputs for Reaction Definition (p. 977) for details.

(Note that the **Pressure-Dependent Reaction** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.)



Controls

Reaction Parameters
contains inputs for specifying the type of pressure fall-off reaction and the reaction parameters. See Pressure-Dependent Reactions in the Theory Guide for details.

Reaction Type
contains a drop-down list of the available reaction types: **lindemann**, **troe**, and **sri**. See Pressure-Dependent Reactions in the Theory Guide for details.

Bath Gas Concentration
allows you to specify if the bath gas concentration ($[M]$ in Equation 7.18 in the Theory Guide) is to be defined as the concentration of the **mixture**, or as the concentration of one of the mixture's constituent species.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chemically Activated Bimolecular Reaction
results in a net rate constant at any pressure being defined as Equation 7.24 in the Theory Guide.

Low Pressure Arrhenius Rate
contains inputs for specifying low-pressure Arrhenius parameters.

ln(Pre-exponential Factor)
is the natural logarithm of the constant A_{low} in Equation 7.16 in the Theory Guide. The pre-exponential factor A_{low} is often an extremely large number, so you will enter the natural logarithm of this term.

Activation Energy
is the constant E_{low} in Equation 7.16 in the Theory Guide.

Temperature Exponent
is the constant β_{low} in Equation 7.16 in the Theory Guide.

Troe parameters
contains inputs for specifying parameters for the Troe method. See Pressure-Dependent Reactions in the Theory Guide for details. (This section of the dialog box will appear only if you have selected **troe** as the **Reaction Type**.)

Alpha
is the constant α in Equation 7.21 in the Theory Guide.

T1
is the constant T_1 in Equation 7.21 in the Theory Guide.

T2
is the constant T_2 in Equation 7.21 in the Theory Guide.

T3
is the constant T_3 in Equation 7.21 in the Theory Guide.

SRI Parameters
contains inputs for specifying parameters for the SRI method. See Pressure-Dependent Reactions in the Theory Guide for details. (This section of the dialog box will appear only if you have selected **sri** as the **Reaction Type**.)

a
is the constant a in Equation 7.22 in the Theory Guide.

b
is the constant b in Equation 7.22 in the Theory Guide.

c
is the constant c in Equation 7.22 in the Theory Guide.

d
is the constant d in Equation 7.22 in the Theory Guide.

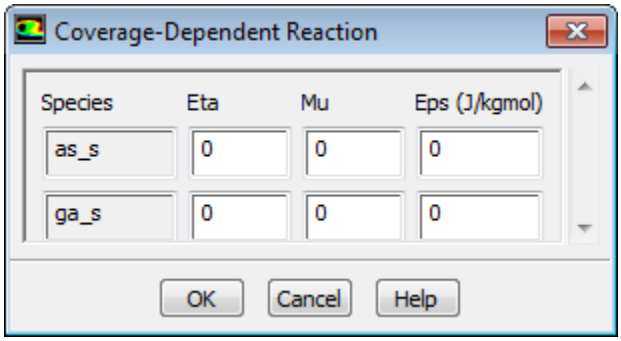
e
is the constant e in Equation 7.22 in the Theory Guide.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.5.31. Coverage-Dependent Reaction Dialog Box

The **Coverage-Dependent Reaction** dialog box (opened by clicking on **Specify...** under **Coverage-Dependent Reaction** in the Reactions Dialog Box (p. 2334)) allows you to model **Wall Surface** reactions with site-balancing.

(Note that the **Coverage-Dependent Reaction** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.)



Controls

Species
are the site species of the reaction.

Eta
is the surface coverage rate modification of the species and is defined in Equation 7.48 in the Theory Guide.

Mu
is the surface coverage rate modification of the species and is defined in Equation 7.48 in the Theory Guide.

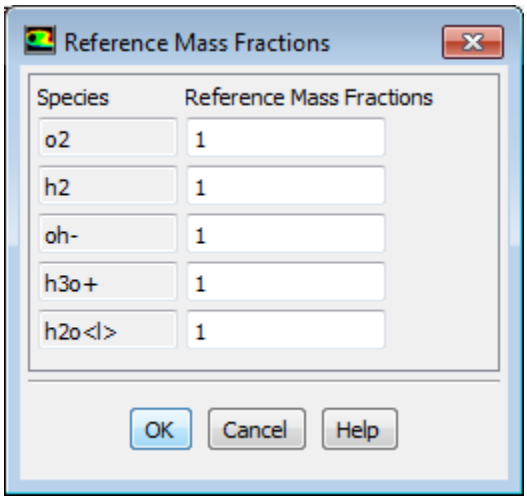
Eps
is the surface coverage rate modification of the species and is defined in Equation 7.48 in the Theory Guide.

37.5.32. Reference Mass Fractions Dialog Box

The **Reference Mass Fractions** dialog box (opened by clicking **Specify Reference Mass Fractions...** under **Butler-Volmer/Tafel Parameters** in the Reactions Dialog Box (p. 2334)) allows you to specify reference species mass fractions for the electrochemical reaction.

(Note that the **Reference Mass Fractions** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Species

are the electrochemical reaction species used in the Butler-Volmer equation (Equation 7.80 in the *Fluent Theory Guide*).

Reference Mass Fractions

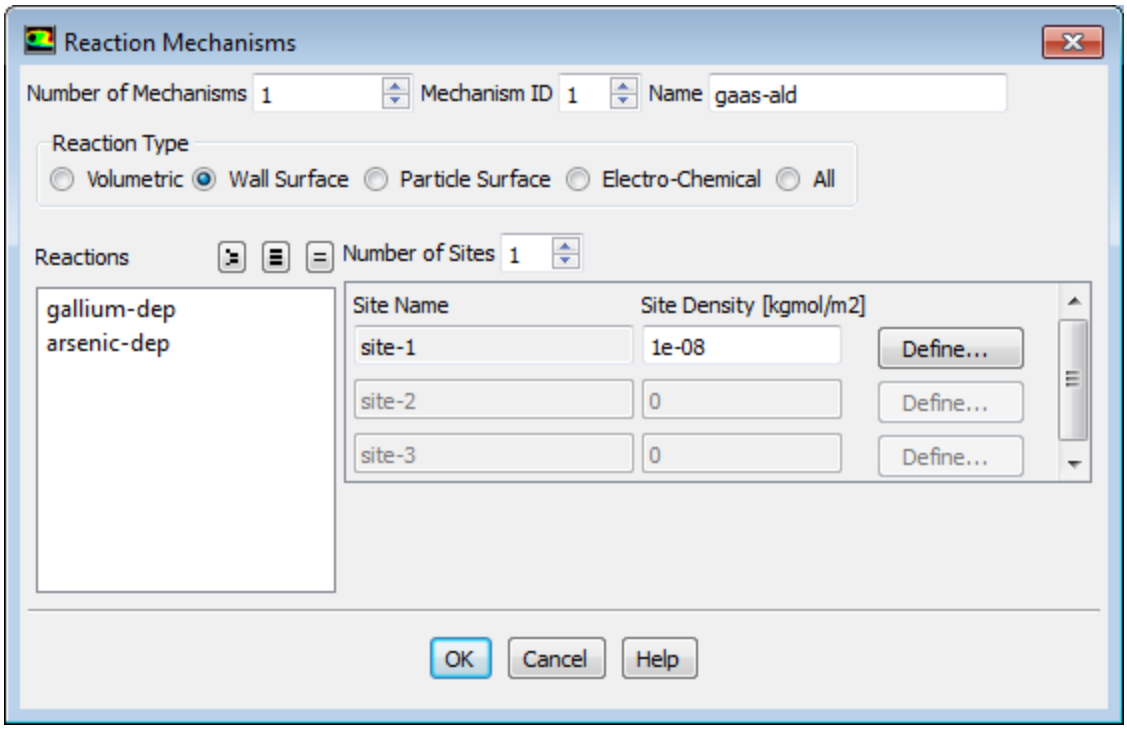
contains inputs for the reference mass fractions of electrochemical reaction species (Y_k in Equation 7.80).

37.5.33. Reaction Mechanisms Dialog Box

The **Reaction Mechanisms** dialog box (opened by clicking the **Edit...** button next to **Mechanism** in the Create/Edit Materials Dialog Box (p. 2302)) allows you to select the reaction mechanism at a particular zone. See Mixture Materials (p. 959) for details about these methods and the related inputs.

(Note that the **Reaction Mechanisms** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Number of Mechanisms
specifies the number of mechanisms present.

Mechanism ID
is the ID of the mechanism that you specify.

Name
allows you to enter a name for the mechanism.

Reaction Type
specifies the type of reaction to be displayed for the mechanism.

Volumetric
displays all volumetric reactions under the **Reactions** list.

Wall Surface
displays all wall surface reactions under the **Reactions** list.

Particle Surface
displays all particle surface reactions under the **Reactions** list.

Electrochemical
displays all electrochemical reactions under the **Reactions** list.

All
displays all types of reactions under the **Reactions** list.

Reactions
displays the list of reactions of the category specified under **Reaction Type**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number of Sites
specifies the number of sites at which you can specify the reaction.

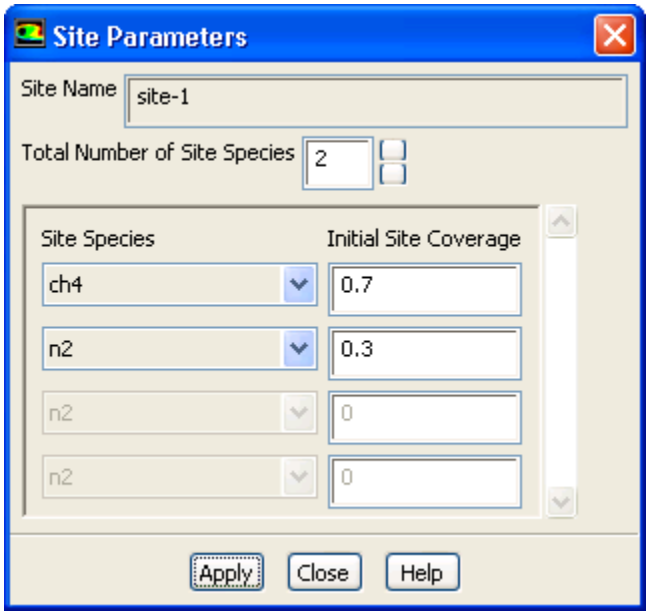
Site Name
contains the name of the site.

Site Density
allows you to specify the site density of the species

Define...
opens the Site Parameters Dialog Box (p. 2344).

37.5.34. Site Parameters Dialog Box

The **Site Parameters** dialog box (opened by clicking the **Define...** button next to **Site Density** in the Reaction Mechanisms Dialog Box (p. 2342)) allows you to define the coverage for each site species.



Controls

Site Name
displays the name of site.

Total Number of Site Species
specifies the total number of site species.

Site Species
allows you to select the site species.

Initial Site Coverage
allows you to specify the initial coverage of the site species.

37.5.35. Mass Diffusion Coefficients Dialog Box

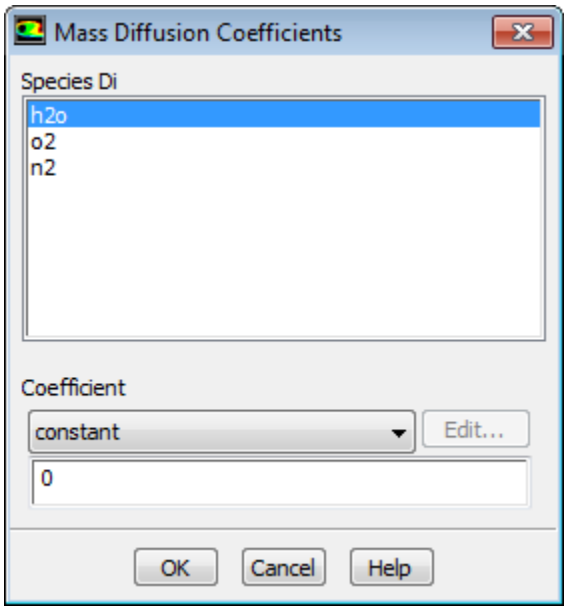
The **Mass Diffusion Coefficients** dialog box (opened by clicking the **Edit...** button next to **Mass Diffusivity** in the Create/Edit Materials Dialog Box (p. 2302) allows you to define the diffusion coefficients of

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the species in the mixture. Its contents will depend on the method you selected for **Mass Diffusivity**. See Mass Diffusion Coefficient Inputs (p. 524) for details about these methods and the related inputs.

(Note that the **Mass Diffusion Coefficients** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.)

For the **dilute-approx** method:



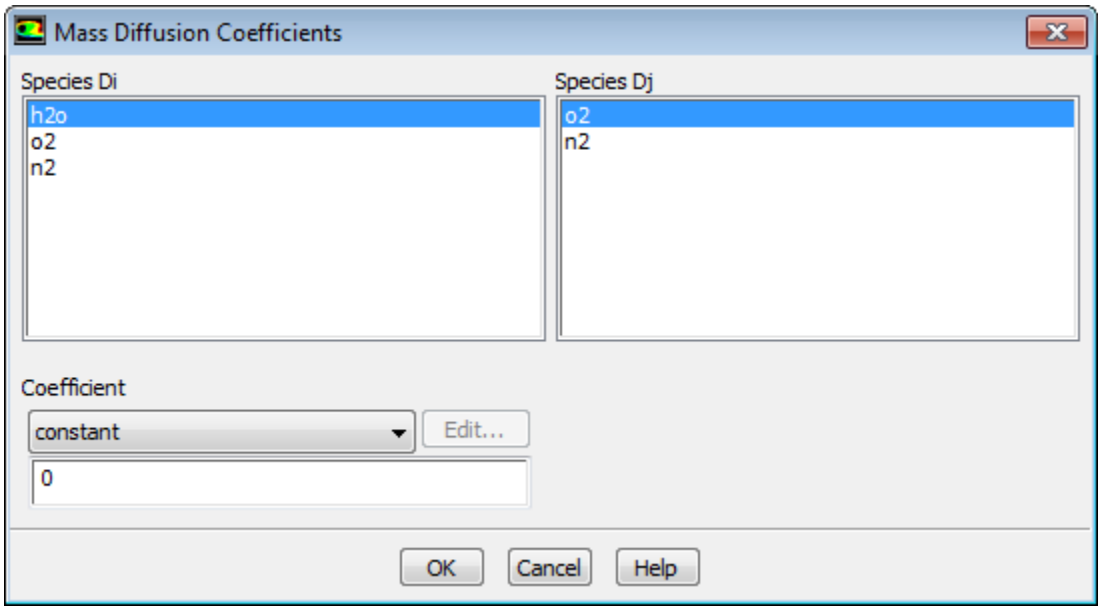
Controls

Species Di
contains a selectable list of all species in the mixture, from which you can select each species and specify its diffusion coefficient.

Coefficient
sets the diffusion coefficient for the selected species in the mixture.

For the **multicomponent** method:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Species Di, Species Dj
contain selectable lists of species in the mixture, from which you can select each pair of species and specify the diffusion coefficient of the selected **Species Di** in the selected **Species Dj**.

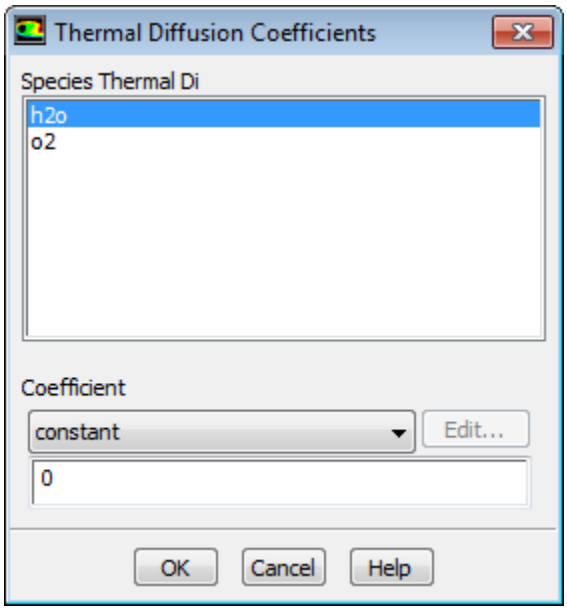
Coefficient
sets the diffusion coefficient for **Species Di** in **Species Dj** (which is equivalent to the diffusion coefficient for **Species Dj** in **Species Di**).

37.5.36. Thermal Diffusion Coefficients Dialog Box

The **Thermal Diffusion Coefficients** dialog box (opened by clicking the **Edit...** button next to **Thermal Diffusion Coefficient** in the Create/Edit Materials Dialog Box (p. 2302) allows you to define the thermal diffusion coefficients of the species in the mixture. See Thermal Diffusion Coefficient Inputs (p. 523) for details about these methods and the related inputs.

(Note that the **Thermal Diffusion Coefficients** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Species Thermal Di
contains a selectable list of all species in the mixture, from which you can select each species and specify its thermal diffusion coefficient.

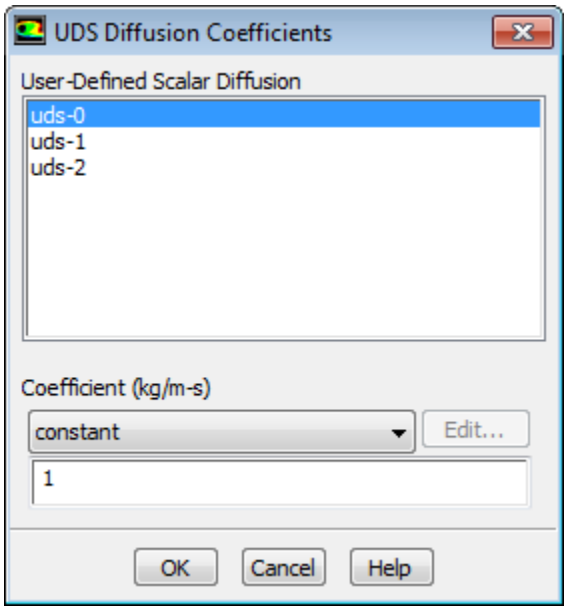
Coefficient
sets the thermal diffusion coefficient for the selected species in the mixture.

37.5.37. UDS Diffusion Coefficients Dialog Box

The **UDS Diffusion Coefficients** dialog box (opened by selecting **uds** and clicking the **Edit...** button next to **UDS Diffusivity** in the Create/Edit Materials Dialog Box (p. 2302)) allows you to define the diffusion coefficients for your user-defined scalar transport equations.

(Note that the **UDS Diffusion Coefficients** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



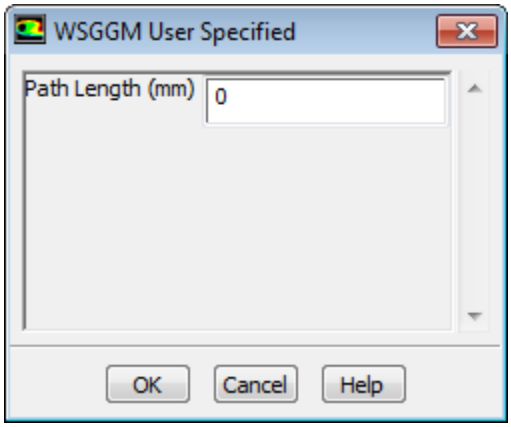
Controls

User-Defined Scalar Diffusion
contains a selectable list of all user-defined scalars, from which you can select each one and specify its diffusion coefficient.

Coefficient
sets the diffusion coefficient for the selected user-defined scalar.

37.5.38. WSGGM User Specified Dialog Box

The **WSGGM User Specified** dialog box opens when you select **wsggm-user-specified** as the input method for a composition-dependent **Absorption Coefficient** in the Create/Edit Materials Dialog Box (p. 2302) and it allows you to define the path length for he weighted-sum-of-gray-gases model. See Inputs for a Composition-Dependent Absorption Coefficient (p. 516) for details.



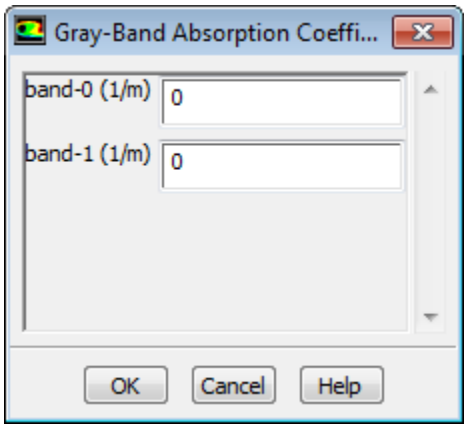
Controls

Path Length
allows you to set the **Path Length** equal to a mean beam length that you have calculated outside of Fluent.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.5.39. Gray-Band Absorption Coefficient Dialog Box

The **Gray-Band Absorption Coefficient** allows you to specify a different absorption coefficient in each gray band when you are modeling non-gray radiation with the P-1, DO, or MC models (see The P-1 Model Equations or The DO Model Equations in the Theory Guide and Setting Up the P-1 Model with Non-Gray Radiation (p. 847) or Defining Non-Gray Radiation for the DO Model (p. 865)). This dialog box will open when you select **gray-band** in the drop-down list next to **Absorption Coefficient** in the Create/Edit Materials Dialog Box (p. 2302).



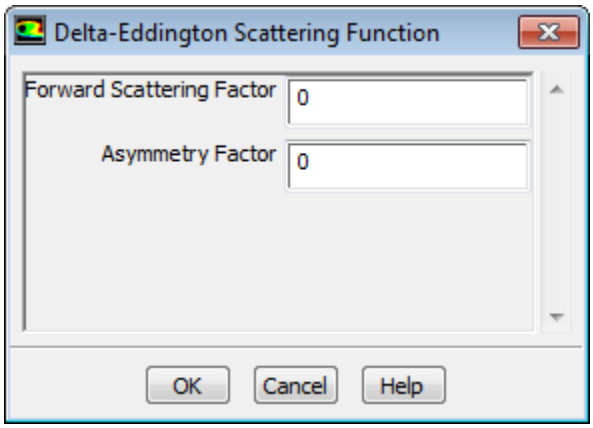
Controls

band n

specifies the absorption coefficient for the n^{th} gray band.

37.5.40. Delta-Eddington Scattering Function Dialog Box

The **Delta-Eddington Scattering Function** dialog box allows you to define the parameters used in the Delta-Eddington phase function for radiation scattering. This dialog box will open when you select **delta-eddington** in the drop-down list next to **Scattering Phase Function** in the Create/Edit Materials Dialog Box (p. 2302). See Anisotropic Scattering for details about the items below.



Controls

Forward Scattering Factor

specifies the value of f in Equation 5.68 in the Theory Guide.

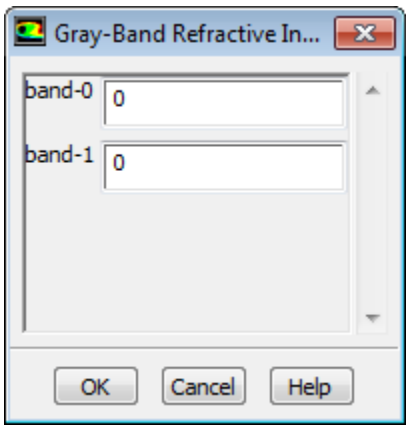
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Asymmetry Factor

specifies the value of C in Equation 5.68 in the Theory Guide.

37.5.41. Gray-Band Refractive Index Dialog Box

The **Gray-Band Refractive Index** allows you to specify a different refractive index in each gray band when you are modeling non-gray radiation with the P-1, DO, or MC model (see The P-1 Model Equations or The DO Model Equations in the Theory Guide and Setting Up the P-1 Model with Non-Gray Radiation (p. 847) or Defining Non-Gray Radiation for the DO Model (p. 865)). This dialog box will open when you select **refractive-band** in the drop-down list next to **Refractive Index** in the Create/Edit Materials Dialog Box (p. 2302).



Controls

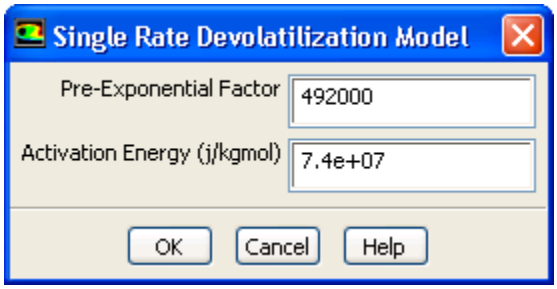
Band n

specifies the refractive index for the n^{th} gray band.

37.5.42. Single Rate Devolatilization Dialog Box

The **Single Rate Devolatilization Model** dialog box (which opens when you select **single-rate** as the **Devolatilization** in the Create/Edit Materials Dialog Box (p. 2302) allows you to specify the parameters used in the single kinetic rate devolatilization model. See Devolatilization (Law 4) in the Theory Guide for details.

Note that the **Single Rate Devolatilization Model** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.



Controls

Pre-exponential Factor

sets the value of A_1 in Equation 16.128 in the Theory Guide for the computation of the kinetic rate.

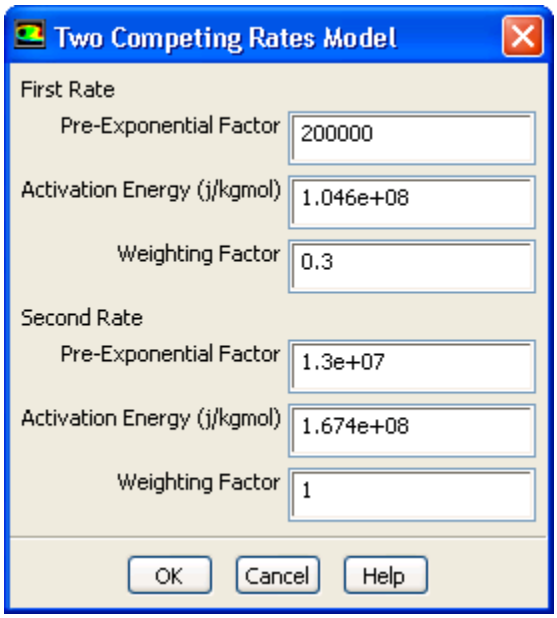
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Activation Energy
sets the value of E in Equation 16.128 in the Theory Guide for the computation of the kinetic rate.

37.5.43. Two Competing Rates Model Dialog Box

The **Two Competing Rates Model** dialog box (which opens when you select **two-competing-rates** as the **Devolatilization Model** in the Create/Edit Materials Dialog Box (p. 2302) allows you to specify the parameters used for each of the competing rates in the two-competing-rates devolatilization model. See Devolatilization (Law 4) in the Theory Guide for details.

Note that the **Two Competing Rates Model** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.



Controls

First Rate
sets parameters for the first of the two rates.

Pre-exponential Factor
sets the value of A_1 in Equation 16.130 in the Theory Guide for the computation of the kinetic rate.

Activation Energy
sets the value of E_1 in Equation 16.130 in the Theory Guide for the computation of the kinetic rate.

Weighting Factor
sets the value of α_1 in Equation 16.132 in the Theory Guide.

Second Rate
sets parameters for the second of the two rates.

Pre-exponential Factor
sets the value of A_2 in Equation 16.131 in the Theory Guide for the computation of the kinetic rate.

Activation Energy
sets the value of E_2 in Equation 16.131 in the Theory Guide for the computation of the kinetic rate.

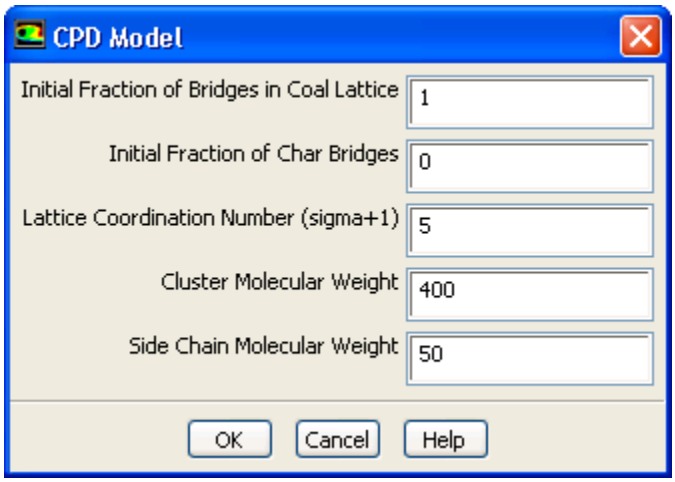
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Weighting Factor
sets the value of α_2 in Equation 16.132 in the Theory Guide.

37.5.44. CPD Model Dialog Box

The **CPD Model** dialog box (which opens when you select **cpd-model** as the **Devolatilization Model** in the Create/Edit Materials Dialog Box (p. 2302)) allows you to specify the parameters used in the CPD devolatilization model. See Devolatilization (Law 4) in the Theory Guide for details.

Note that the **CPD Model** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.



Controls

Initial Fraction of Bridges in Coal Lattice
sets the value of p_0 in Equation 16.143 in the Theory Guide.

Initial Fraction of Char Bridges
sets the value of c_0 in Equation 16.142 in the Theory Guide.

Lattice Coordination Number
sets the value of $\sigma+1$ in Equation 16.154 in the Theory Guide.

Cluster Molecular Weight
sets the value of $M_{w,1}$ in Equation 16.154 in the Theory Guide.

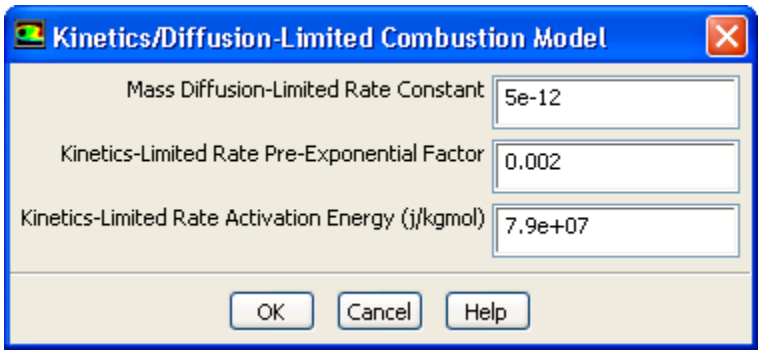
Side Chain Molecular Weight
sets the value of $M_{w,\delta}$ in Equation 16.153 in the Theory Guide.

37.5.45. Kinetics/Diffusion-Limited Combustion Model Dialog Box

The **Kinetics/Diffusion-Limited Combustion Model** dialog box (which opens when you select **kinetics/diffusion-limited** as the **Combustion Model** in the Create/Edit Materials Dialog Box (p. 2302) allows you to specify the parameters used for the kinetics/diffusion-limited rate surface combustion model. See Surface Combustion (Law 5) in the Theory Guide for details.

Note that the **Kinetics/Diffusion-Limited Combustion Model** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Mass Diffusion-Limited Rate Constant
sets the value for C_1 in Equation 16.166 in the Theory Guide.

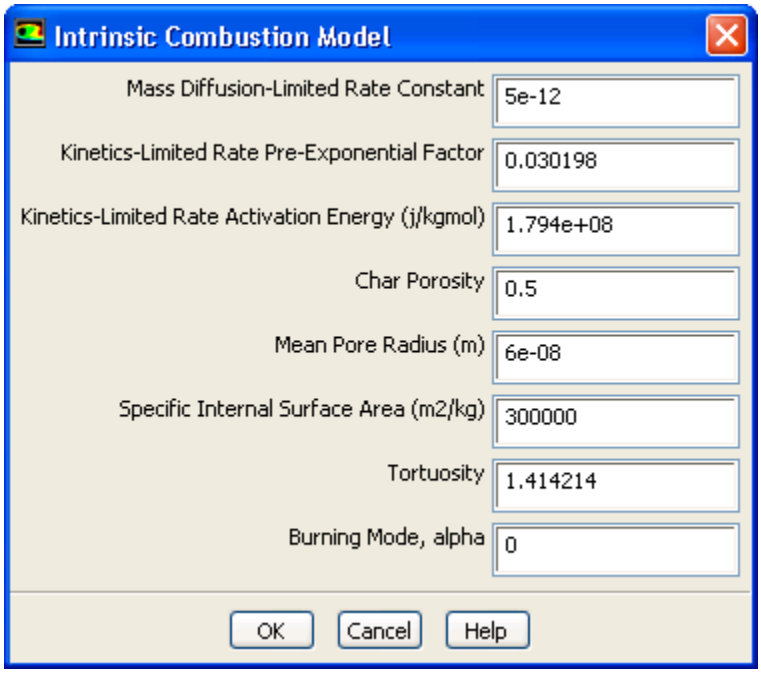
Kinetics-Limited Rate Pre-exponential Factor
sets the value for C_2 in Equation 16.167 in the Theory Guide.

Kinetics-Limited Rate Activation Energy
sets the value for E in Equation 16.167 in the Theory Guide.

37.5.46. Intrinsic Combustion Model Dialog Box

The **Intrinsic Combustion Model** dialog box (which opens when you select **intrinsic-model** as the **Combustion Model** in the Create/Edit Materials Dialog Box (p. 2302) allows you to specify the parameters used for the intrinsic surface combustion model. See Surface Combustion (Law 5) in the Theory Guide for details.

Note that the **Intrinsic Combustion Model** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.



Controls

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mass Diffusion-Limited Rate Constant
sets the value for C_1 in Equation 16.166 in the Theory Guide.

Kinetics-Limited Rate Pre-exponential Factor
sets the value for A_i in Equation 16.176 in the Theory Guide.

Kinetics-Limited Rate Activation Energy
sets the value for E_i in Equation 16.176 in the Theory Guide.

Char Porosity
sets the value for θ in Equation 16.173 in the Theory Guide.

Mean Pore Radius
sets the value for $\overline{r}_p$ in Equation 16.175 in the Theory Guide.

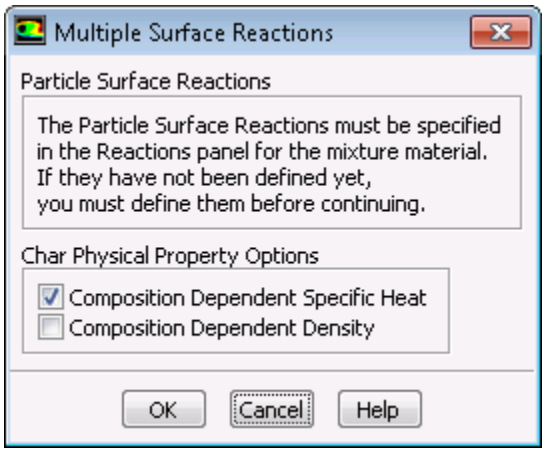
Specific Internal Surface Area
sets the value for A_g in Equation 16.170 and Equation 16.172 in the Theory Guide.

Tortuosity
sets the value for τ in Equation 16.173 in the Theory Guide.

Burning Mode, alpha
sets the value for α in Equation 16.177 in the Theory Guide.

37.5.47. Multiple Surface Reactions Dialog Box

The **Multiple Surface Reactions** dialog box (which opens when you select **multiple-surface-reactions** as the **Combustion Model** in the Create/Edit Materials Dialog Box (p. 2302)) allows you to enable composition-dependent char properties when using the multiple surface reactions combustion model.



Controls

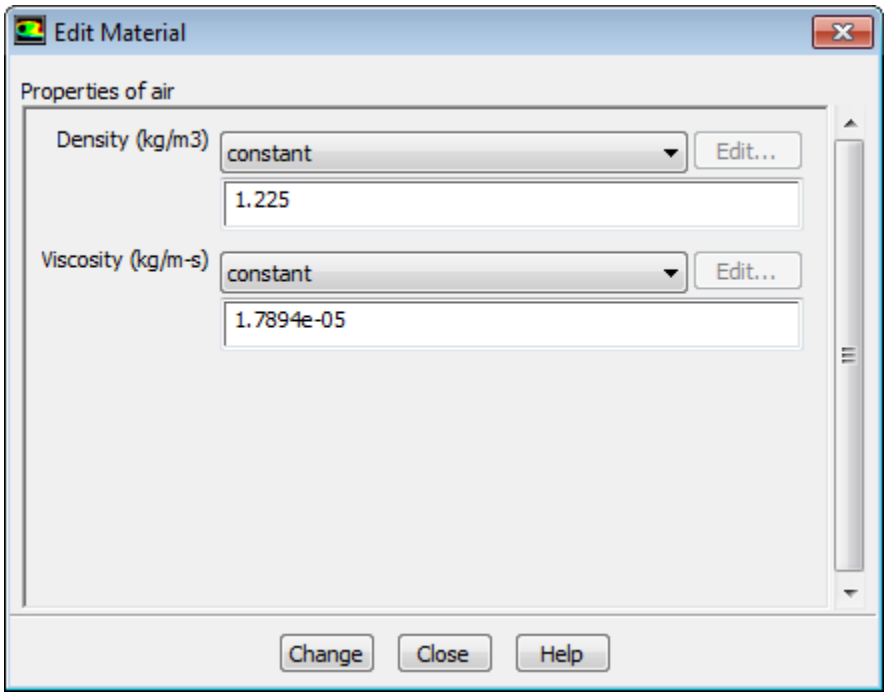
Composition Dependent Specific Heat
enables calculation of the char specific heat from the particle specific heat values. See **Combustion Model** (p. 1319) for details.

Composition Dependent Density
enables calculation of the char density from the particle density values. See **Combustion Model** (p. 1319) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.5.48. Edit Material Dialog Box

The **Edit Material** dialog box contains the portion of the Create/Edit Materials Dialog Box (p. 2302) that contains the properties for a specific material. It is opened from the Primary Phase Dialog Box (p. 2357), Secondary Phase Dialog Box (p. 2357), Wall Dialog Box (p. 2456), Fluid Dialog Box (p. 2369), or Solid Dialog Box (p. 2379).



Controls

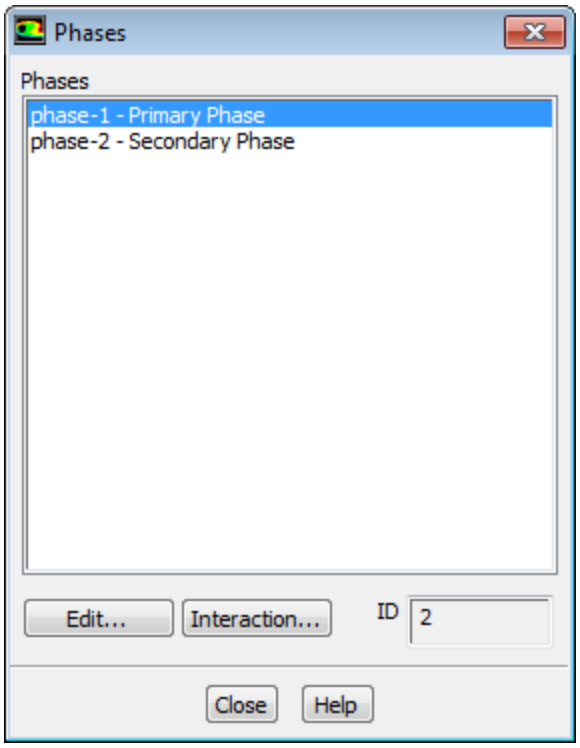
Properties of material-n
contains a list of the properties of material-*n*. The items in the list are the same as those in the Create/Edit Materials Dialog Box (p. 2302).

Change
applies any changes you have made to the properties of the material.

37.6. Phases

The **Phases** dialog box allows you to define each of the phases and the interaction between them. See – Defining the Phases for the Eulerian Model (p. 1456) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Phases

contains a list of all of the phases in the problem from which you can select the phase you want to define or modify. A phase can be a **Primary Phase** or a **Secondary Phase**. You cannot change a phase from primary to secondary, or vice versa. Instead, you can redefine the properties of the primary phase to reflect the new phase designated as primary, and redefine the secondary phases accordingly as well.

Edit...

opens either the Primary Phase Dialog Box (p. 2357) or the Secondary Phase Dialog Box (p. 2357), where you can define the properties of the selected primary or secondary phase.

Interaction...

opens the Phase Interaction Dialog Box (p. 2363), where you can define the interaction between the phases (for example, surface tension if you are using the VOF model, slip velocity functions if you are using the mixture model, or drag functions if you are using the Eulerian model).

ID

displays the ID number of the phase. You will need this number only if you are writing a user-defined function. See the separate Fluent Customization Manual for details about writing user-defined functions for multiphase applications.

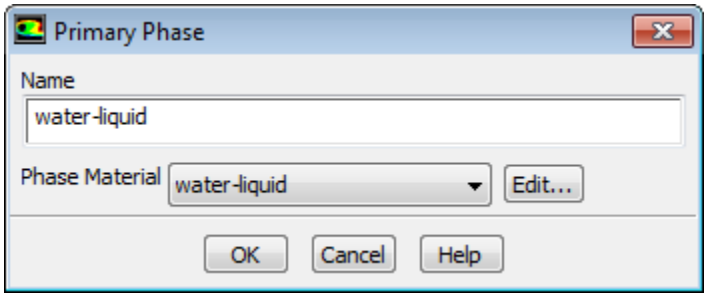
For additional information, see the following sections:

37.6.1. Primary Phase Dialog Box
37.6.2. Secondary Phase Dialog Box
37.6.3. Discrete Phase Dialog Box
37.6.4. Phase Interaction Dialog Box

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.6.1. Primary Phase Dialog Box

The **Primary Phase** dialog box allows you to set the properties of the primary phase. It is opened from the Phases (p. 2355). See Defining the Phases for the VOF Model (p. 1436) for details about the items below.



Controls

Name
specifies the name of the phase.

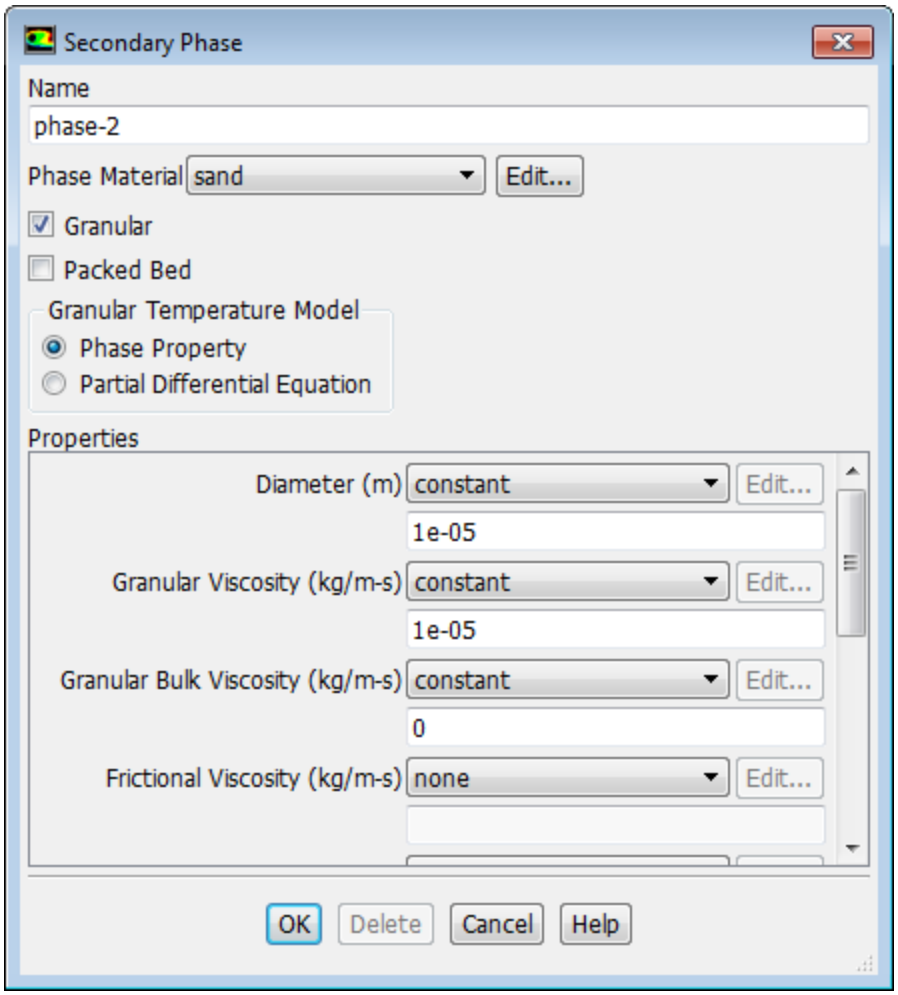
Phase Material
contains a drop-down list of available materials, from which you can select the appropriate one for this phase.

Edit...
opens the Edit Material Dialog Box (p. 2355) for the selected **Phase Material**, where you can modify its properties.

37.6.2. Secondary Phase Dialog Box

The **Secondary Phase** dialog box allows you to set the properties of a secondary phase. It is opened from the Phases (p. 2355). The items that appear in the **Secondary Phase** dialog box will depend on which multiphase model you are using. See Defining the Phases for the VOF Model (p. 1436), Defining the Phases for the Mixture Model (p. 1447), and Defining the Phases for the Eulerian Model (p. 1456) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name
specifies the name of the phase.

Phase Material
contains a drop-down list of available materials, from which you can select the appropriate one for the phase.

Edit...
opens the Edit Material Dialog Box (p. 2355) for the selected **Phase Material**, where you can modify its properties.

Granular
indicates whether or not this is a solid phase. This item appears only for the Eulerian model.

Packed Bed
indicates whether or not the granular phase is a packed bed. This option appears only if **Granular** is enabled.

Granular Temperature Model
lists the granular temperature models.

Phase Property
enables phase property model for granular temperature.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Partial Differential Equation

enables partial differential equation model for granular temperature. See Granular Temperature in the Theory Guide for details.

Interfacial Area Concentration

is used to predict mass, momentum and energy transfers through the interface between the phases. See Interfacial Area Concentration in the Theory Guide for details.

Properties

contains a list of phase-specific properties. This section of the dialog box will not appear for the VOF model. The **Diameter** appears for both the mixture model and the Eulerian model, but all of the others will appear only for a granular phase with the Eulerian model.

Diameter

specifies the diameter of the particles. You can select **constant** in the drop-down list and specify a constant value, or select **user-defined** to use a user-defined function. See the separate Fluent Customization Manual for details about user-defined functions.

Granular Viscosity

specifies the kinetic part of the granular viscosity of the particles ($\mu_{s,kin}$ in Equation 17.307 in the Theory Guide). You can select **constant** (the default) in the drop-down list and specify a constant value, select **syamlal-obrien** to compute the value using Equation 17.309 in the Theory Guide, select **gidaspow** to compute the value using Equation 17.310 in the Theory Guide, or select **user-defined** to use a user-defined function.

Granular Bulk Viscosity

specifies the solids bulk viscosity (λ_q in Equation 17.169 in the Theory Guide). You can select **constant** (the default) in the drop-down list and specify a constant value, select **lun-et-al** to compute the value using Equation 17.311 in the Theory Guide, or select **user-defined** to use a user-defined function.

Frictional Viscosity

specifies a shear viscosity based on the viscous-plastic flow ($\mu_{s,fr}$ in Equation 17.307 in the Theory Guide). By default, the frictional viscosity is neglected, as indicated by the default selection of **none** in the drop-down list. If you want to include the frictional viscosity, you can select **constant** and specify a constant value, select **schaeffer** to compute the value using Equation 17.312 in the Theory Guide, **johnson-et-al**, or select **user-defined** to use a user-defined function.

Angle Of Internal Friction

specifies a constant value for the angle φ used in Schaeffer's expression for frictional viscosity (Equation 17.312 in the Theory Guide). This parameter is relevant only if you have selected **schaeffer**, **johnson-et-al**, or **user-defined** for the **Frictional Viscosity**.

Frictional Pressure

specifies the pressure gradient term, $\nabla P_{friction}$, in the granular-phase momentum equation. Choose **none** to exclude frictional pressure from your calculation, **johnson-et-al** to apply Equation 17.317 in the Theory Guide, **syamlal-et-al** to apply Equation 17.228 in the Theory Guide, **based-ktgf**, where the frictional pressure is defined by the kinetic theory [26] (p. 2888). The solids pressure tends to a large value near the packing limit, depending on the model selected for the radial distribution function. You must hook a user-defined function when selecting the **user-defined** option. See the separate Fluent Customization Manual for information on hooking a UDF.

Frictional Modulus

can be set as **derived**, or as a **user-defined** function. This is defined as Equation 24.9 (p. 1450).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Friction Packing Limit
specifies a threshold volume fraction at which the frictional regime becomes dominant. It is assumed that for a maximum packing limit of 0.6, the frictional regime starts at a volume fraction of about 0.5.

Granular Conductivity
specifies the solids conductivity. You can select **syamlal-obrien**, **gidaspow**, **constant** or **user-defined**.

Granular Temperature
specifies temperature for the solids phase and is proportional to the kinetic energy of the random motion of the particles. You can choose the **algebraic**, **constant**, **dpm-averaged**,or **user-defined** option. **dpm-averaged** is available only when using the Dense Discrete Phase Model (DDPM).

Solids Pressure
specifies the pressure gradient term, ∇p_s, in the granular-phase momentum equation. Choose either the **lun-et-al**, the **syamlal-obrien**, the **ma-ahmadi**, or the **user-defined** option.

Radial Distribution
specifies a correction factor that modifies the probability of collisions between grains when the solid granular phase becomes dense. Choose either the **lun-et-al**, the **syamlal-obrien**, the **ma-ahmadi**, the **arastapour**, or a **user-defined** option.

Elasticity Modulus
is defined as

$$G=\frac{\partial P_s}{\partial \alpha_s} \tag{37.1}$$

with $G \geq 0$.

Choose either the **derived** or **user-defined** options.

Packing Limit
specifies the maximum volume fraction for the granular phase ($\alpha_{s,max}$). For monodispersed spheres the packing limit is about 0.63, which is the default value in ANSYS Fluent. In polydispersed cases, however, smaller spheres can fill the small gaps between larger spheres, so you may need to increase the maximum packing limit.

Surface Tension
specifies the attractive forces between the interfaces.

Coalescence Kernel
allows you to specify the coalescence kernel. You can select **none**, **constant**, **hibiki-ishii**, **ishii-kim**, **yao-morel**, or **user-defined**. The three options, **hibiki-ishii**, **ishii-kim**, and **yao-morel** are described in detail in Interfacial Area Concentration in the Theory Guide.

Breakage Kernel
allows you to specify the breakage kernel. You can select **none**, **constant**, **hibiki-ishii**, **ishii-kim**, **yao-morel**, or **user-defined**. The three options, **hibiki-ishii**, **ishii-kim**, and **yao-morel** are described in detail in Interfacial Area Concentration in the Theory Guide.

Nucleation Rate
is a source term for the interfacial area concentration that models the rate of formation of the dispersed phase. You can choose from **constant** or **user-defined**. If the **Boiling Model** option is enabled when using the Eulerian multiphase model, you can also select **yao-morel**. The **yao-morel** option is described in Yao-Morel Model in the Theory Guide.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Critical Weber Number
will need to be specified if you selected **yao-morel** for the **Breakage Kernel**.

Dissipation Function
gives you the option to choose the formula which calculates the dissipation rate used in the **hibiki-ishii** and **ishii-kim** models. You can choose **constant**, **wu-ishii-kim**, **fluent-ke**, and **user-defined** for the dissipation function.

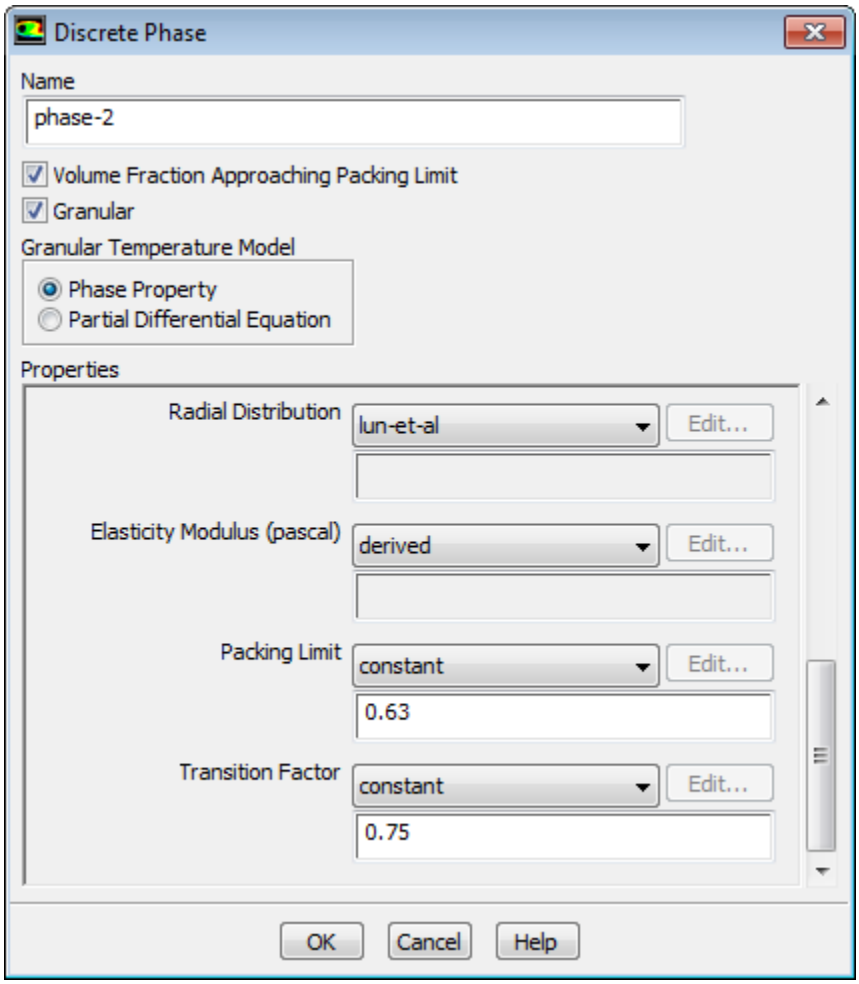
Hydraulic Diameter
is the value used in Equation 24.11 (p. 1453). This is available when the **wu-ishii-kim** formulation is selected as the **Dissipation Function**.

Min/Max Diameter
are the limits of the bubble diameters.

Delete
deletes this phase.

37.6.3. Discrete Phase Dialog Box

The **Discrete Phase** dialog box allows you to set the properties of a discrete phase. It is opened from the Phases (p. 2355). The items will appear in the **Discrete Phase** dialog box when the **Dense Discrete Phase Model** option is enabled in the **Multiphase Model** dialog box. See Including the Dense Discrete Phase Model (p. 1481) for details about the items below.



Controls

Name
specifies the name of the phase.

Volume Fraction Approaching Continuous Flow Limit
appears for non-granular flows. When this option is enabled, only the **Transition Factor** must be specified.

Volume Fraction Approaching Packing Limit
prevents the unlimited accumulation of particles, which are operating at packing limit conditions.

Granular
indicates whether or not this is a solid phase. This item appears only for the Eulerian model.

Granular Temperature Model
lists the granular temperature models.

Phase Property
enables phase property model for granular temperature.

Partial Differential Equation
enables partial differential equation model for granular temperature. See Granular Temperature in the Theory Guide for details.

Properties
contains a list of phase-specific properties.

Granular Viscosity
specifies the kinetic part of the granular viscosity of the particles ($\mu_{s,kin}$ in Equation 17.307 in the Theory Guide). You can select **constant** (the default) in the drop-down list and specify a constant value, select **syamlal-obrien** to compute the value using Equation 17.309 in the Theory Guide, select **gidaspow** to compute the value using Equation 17.310 in the Theory Guide, or select **user-defined** to use a user-defined function.

Granular Bulk Viscosity
specifies the solids bulk viscosity (λ_q in Equation 17.169 in the Theory Guide). You can select **constant** (the default) in the drop-down list and specify a constant value, select **lun-et-al** to compute the value using Equation 17.311 in the Theory Guide, or select **user-defined** to use a user-defined function.

Frictional Viscosity
specifies a shear viscosity based on the viscous-plastic flow ($\mu_{s,fr}$ in Equation 17.307 in the Theory Guide). By default, the frictional viscosity is neglected, as indicated by the default selection of **none** in the drop-down list. If you want to include the frictional viscosity, you can select **constant** and specify a constant value, select **schaeffer** to compute the value using Equation 17.312 in the Theory Guide, **johnson-et-al**, or select **user-defined** to use a user-defined function.

Angle Of Internal Friction
specifies a constant value for the angle φ used in Schaeffer's expression for frictional viscosity (Equation 17.312 in the Theory Guide). This parameter is relevant only if you have selected **schaeffer**, **johnson-et-al**, or **user-defined** for the **Frictional Viscosity**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Frictional Pressure

specifies the pressure gradient term, $\nabla P_{friction}$, in the granular-phase momentum equation. Choose **none** to exclude frictional pressure from your calculation, **johnson-et-al** to apply Equation 17.317 in the Theory Guide, **syamlal-obrien** to apply Equation 17.228 in the Theory Guide, **based-ktgf**, where the frictional pressure is defined by the kinetic theory [26] (p. 2888). The solids pressure tends to a large value near the packing limit, depending on the model selected for the radial distribution function. You must hook a user-defined function when selecting the **user-defined** option. See the separate Fluent Customization Manual for information on hooking a UDF.

Frictional Modulus

can be set as **derived**, or as a **user-defined** function. This is defined as Equation 24.9 (p. 1450).

Friction Packing Limit

specifies a threshold volume fraction at which the frictional regime becomes dominant. It is assumed that for a maximum packing limit of 0.6, the frictional regime starts at a volume fraction of about 0.5.

Granular Conductivity

specifies the solids conductivity. You can select **syamlal-obrien**, **gidaspow**,or **user-defined**.

Solids Pressure

specifies the pressure gradient term, ∇p_s, in the granular-phase momentum equation. Choose either the **lun-et-al**, the **syamlal-obrien**, the **ma-ahmadi**, or the **user-defined** option.

Radial Distribution

specifies a correction factor that modifies the probability of collisions between grains when the solid granular phase becomes dense. Choose either the **lun-et-al**, the **syamlal-obrien**, the **ma-ahmadi**, the **arastapour**, or a **user-defined** option.

Elasticity Modulus

is defined as

$$G=\frac{\partial P_s}{\partial \alpha_s} \tag{37.2}$$

with $G \geq 0$.

Choose either the **derived** or **user-defined** options.

Packing Limit

specifies the maximum volume fraction for the granular phase ($\alpha_{s,max}$). For monodispersed spheres the packing limit is about 0.63, which is the default value in ANSYS Fluent. In polydispersed cases, however, smaller spheres can fill the small gaps between larger spheres, so you may need to increase the maximum packing limit.

Transition Factor

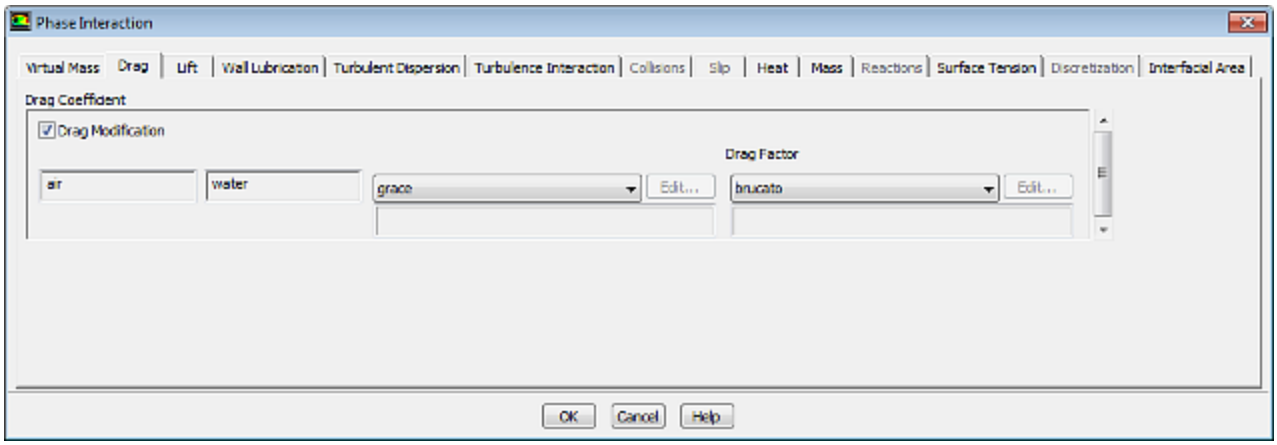
is specified as either a **constant** or a **user-defined** function. The default value is assumed to be 0.75, which corresponds to the closest sphere packing for monosized spheres (a factor of 4/3). In other words, the transition criterion is based on the local particle volume fraction of the given discrete phase and is specified as a factor multiplied by the maximum packing limit (also a user specified value).

37.6.4. Phase Interaction Dialog Box

The **Phase Interaction** dialog box allows you to define the interaction between phases. It is opened from the Phases (p. 2355). The items that appear in the **Phase Interaction** dialog box will depend on

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

which multiphase model you are using. See Defining the Phases for the VOF Model (p. 1436), Defining the Phases for the Mixture Model (p. 1447), and Defining the Phases for the Eulerian Model (p. 1456) for details about the items below.



Controls

Virtual Mass

displays the Virtual Mass Coefficient inputs. This tab is active only for the Eulerian models.

Virtual Mass Modeling

includes the "virtual mass force" (F_{vm} in Equation 17.288 in the Theory Guide) that is present when a secondary phase accelerates relative to the primary phase.

Implicit

enables the implicit method for the virtual mass force which can improve convergence in some cases.

Virtual Mass Implicit Options

specifies what form of the implicit method to use. **Default** models the entire virtual mass force while **Option 2** and **Option 3** model truncated expressions which may further improve convergence.

Virtual Mass Coefficient

specifies the virtual mass coefficient for each pair of phases. See Including the Virtual Mass Force (p. 1474) for details.

Drag

displays the **Drag Coefficient** inputs. This tab is active only for the Eulerian and mixture models.

Drag Coefficient

specifies the drag function for each pair of phases. This section of the dialog box appears only for the Eulerian and mixture models. See Defining the Phases for the Eulerian Model (p. 1456) for information about the available options.

Drag Modification

enables modification of the selected drag models by a drag factor and expands the dialog box to show the **Drag Factor** settings. See Drag Modification (p. 1466) for additional information on using Drag Modification.

Drag Factor

specifies the drag factor to use for modifying the specified drag model. See Drag Modification (p. 1466) for additional information on specifying the **Drag Factor**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lift

displays the **Lift Coefficient** inputs. This tab is active only for the Eulerian model.

Lift Coefficient

specifies the lift function for each pair of phases. This section of the dialog box appears only for the Eulerian model. See Defining the Phases for the Eulerian Model (p. 1456) for information about the available options.

Wall Lubrication

displays the **Wall Lubrication** inputs. This tab is active only for the Eulerian model.

Wall Lubrication

specifies the wall lubrication model for each primary-secondary phase pair. This section of the dialog box appears only for the Eulerian model. See Including the Wall Lubrication Force (p. 1467) for information about the available options.

Turbulent Dispersion

displays the **Turbulent Dispersion** inputs. This tab is active only for the Eulerian model.

Turbulent Dispersion

specifies the turbulent dispersion model for each primary-secondary phase pair. This section of the dialog box appears only for the Eulerian model. See Including the Turbulent Dispersion Force (p. 1470) for information about the available options.

Turbulence Interaction

displays the **Turbulence Interaction** inputs. This tab is active only for the Eulerian model.

Turbulence Interaction

specifies the turbulence interaction model for each primary-secondary phase pair. This section of the dialog box appears only for the Eulerian model. See Including Turbulence Interaction Source Terms (p. 1476) for information about the available options.

Collisions

displays the **Restitution Coefficient** inputs.

Restitution Coefficient

specifies the restitution coefficient for collisions between each pair of granular phases, and for collisions between particles of the same granular phase. It is relevant only if two or more granular phases are involved. See Defining the Phases for the Eulerian Model (p. 1456) for information about the available options.

Slip

displays the **Slip Velocity** inputs. This tab is active only for the mixture model.

Slip Velocity

specifies the slip velocity function for each secondary phase with respect to the primary phase. See Defining the Phases for the Mixture Model (p. 1447) for information about the available options.

Heat

displays the **Heat Transfer Coefficient** inputs. This tab is active only for the Eulerian model when the energy equation is active.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Heat Transfer Coefficient
specifies the heat transfer coefficient function between each pair of phases. See Including Heat Transfer Effects (p. 1478) for information about the available options.

Mass
displays the **Mass Transfer Function** inputs.

Number of Mass Transfer Mechanisms
specifies the number of mass transfer mechanisms in your simulation. See Including Mass Transfer Effects (p. 1383) for information about the available options.

Reactions
allows you to define multiple heterogeneous reactions and stoichiometry.

Total Number of Heterogeneous Reactions
specifies the total number of reactions (volumetric reactions, wall surface reactions, and particle surface reactions). See Specifying Heterogeneous Reactions (p. 1380) for information about the available options.

Heterogeneous Stiff Chemistry Solver
is used in inter-phase reaction mechanisms containing numerically stiff reactions. This option can improve convergence and is available for transient Eulerian multiphase simulations.

Reaction Name
allows you to enter a name for the reaction.

ID
enables you to set the reaction ID for each reaction.

Number of Reactants
allows you to specify the number of reactants that are involved in the reaction.

Number of Products
allows you to specify the number of products that are involved in the reaction.

Phase
drop-down list allows you to select the phase that is involved in the reaction.

Species
drop-down list allows you to select the species.

Stoich. Coefficient
allows you to set the stoichiometric coefficient.

Reaction Rate Function
allows you to choose rate exponents for an Arrhenius-type reaction, a user-defined function, or a population balance mechanism for the reaction rate.

Surface Tension
includes the effects of surface tension along the fluid-fluid interface.

Model
contains two surface tension models from which to choose.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Continuum Surface Force
adds the surface tension to the VOF calculation, which results in a source term in the momentum equation. This method is available only for the VOF and Eulerian models.

Continuum Surface Stress
is an alternative way to modeling surface tension in a conservative manner compared to the continuum surface force method. This method is available only for the VOF and Eulerian models.

Adhesion Options
contains options to include wall and jump adhesion.

Wall Adhesion
enables the specification of a wall adhesion angle. (The angle itself, as defined in Figure 24.31: Measuring the Contact Angle (p. 1442), will be specified in the Wall Dialog Box (p. 2456).) This item will appear only for the VOF and Eulerian models.

Jump Adhesion
enables the treatment of the contact angle specification at the porous jump boundary. (The angle itself will be specified in the Porous Jump Dialog Box (p. 2427).)

Surface Tension Coefficients
specify the surface tension coefficient, σ in Equation 17.22 and Equation 16.80 in the Theory Guide, for each pair of phases. See Defining the Phases for the VOF Model (p. 1436) for details.

Discretization
allows you to use the diffusive and anti-diffusive discretization procedure across the distinct interfaces.

Phase Localized Compressive Scheme
enables the compressive discretization scheme in ANSYS Fluent, where the degree of diffusion/sharpness is controlled through the value of the slope limiters. This item will appear only for the VOF model and for the Eulerian model with **Multi-Fluid VOF Model** enabled.

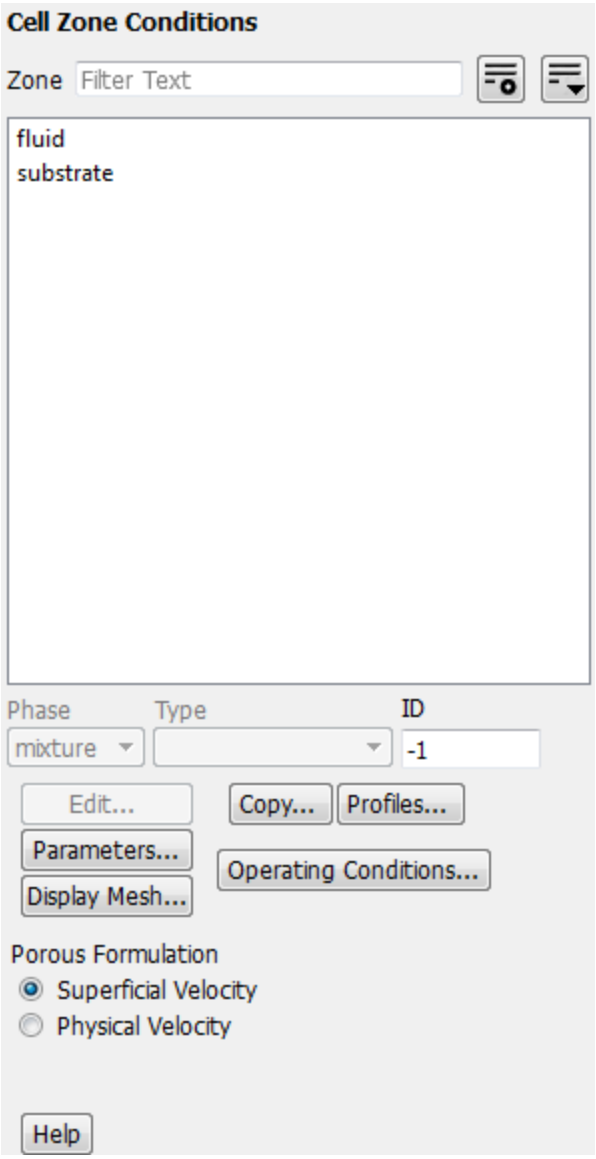
Slope Limiters
are values equating to specific discretization schemes. For each pair of phases, you can enter a value of 0, 1, or 2, or any value between 0 and 2. Depending on the value you use, first order upwind, second order upwind, compressive, or the blended scheme will be applied. Refer to Table 24.11: Slope Limiter Discretization Scheme (p. 1444) to equate each value of the slope limiter with a discretization scheme. For more information, refer to Discretizing Using the Phase Localized Compressive Scheme (p. 1443).

Interfacial Area
displays the **Interfacial Area** inputs. This tab only appears for the Eulerian Model. See Using an Algebraic Interfacial Area Model (p. 1480) and Interfacial Area Concentration in the *Fluent Theory Guide* for information about the available options.

37.7. Cell Zone Conditions Task Page

The **Cell Zones Conditions** task page allows you to set the type of a cell zone and display other dialog boxes to set the cell zone condition parameters for each zone. See Cell Zone Conditions (p. 252) for more information.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone

contains a selectable list of available cell zones from which you can select the zone of interest. You can check a zone type by using the mouse probe (see Controlling the Mouse Button Functions (p. 1800)) on the displayed physical mesh. This feature is particularly useful if you are setting up a problem for the first time, or if you have two or more cell zones of the same type and you want to determine the cell zone IDs. To do this you must first display the mesh with the Mesh Display Dialog Box (p. 2163). Then click the boundary zone with the right (select) mouse button. ANSYS Fluent will print the cell zone ID and type of that boundary zone in the console window.

Phase

specifies the phase for which conditions at the selected cell **Zone** are being set. This item appears if the VOF, mixture, or Eulerian multiphase model is being used. See Defining Multiphase Cell Zone and Boundary Conditions (p. 1391) for details.

Type

contains a drop-down list of condition types for the selected cell zone. The list contains all possible types to which the cell zone can be changed.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ID
displays the cell zone ID number of the selected cell zone. (This is for informational purposes only; you cannot edit this number.)

Edit...
opens the Fluid Dialog Box (p. 2369) or Solid Dialog Box (p. 2379).

Copy...
opens the Copy Conditions Dialog Box (p. 2381), which allows you to copy conditions from one cell zone to other cell zones of the same type. See Copying Cell Zone and Boundary Conditions (p. 242) for details.

Profiles...
opens the Profiles Dialog Box (p. 2385).

Parameters...
opens the Parameters Dialog Box (p. 2627).

Operating Conditions...
opens the Operating Conditions Dialog Box (p. 2382).

Display Mesh...
opens the Mesh Display Dialog Box (p. 2163).

Porous Formulation
contains options for setting the velocity in the porous medium simulation. See Defining the Porous Velocity Formulation (p. 269) for details.

Superficial Velocity
enables the superficial velocity in a porous medium simulation. This is the default method.

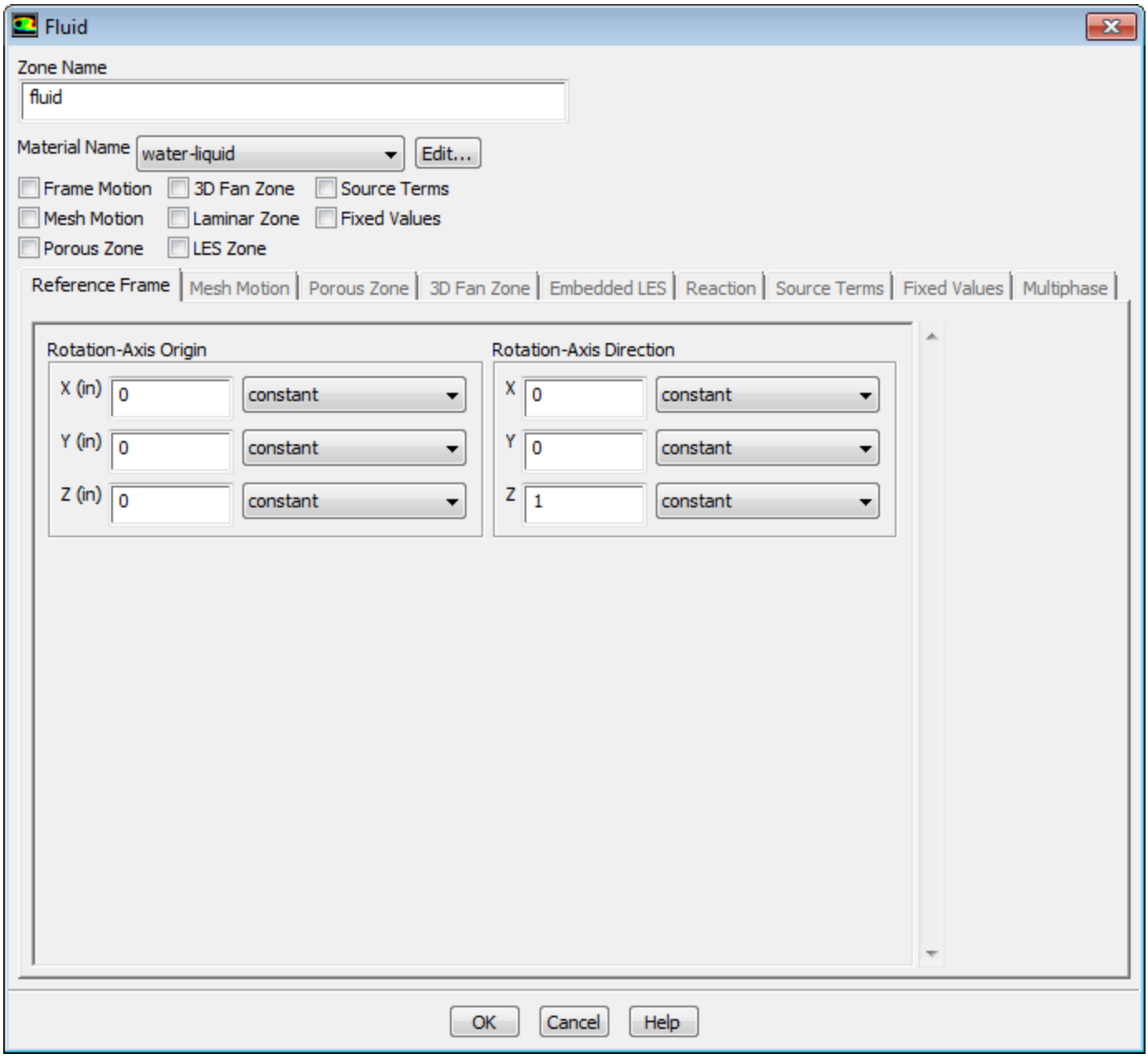
Physical Velocity
enables the physical velocity in a porous medium simulation for a more accurate simulation. This option is available only for a pressure-based solver. See Modeling Porous Media Based on Physical Velocity (p. 283) for details.

For additional information, see the following sections:
37.7.1. Fluid Dialog Box
37.7.2. Solid Dialog Box
37.7.3. Copy Conditions Dialog Box
37.7.4. Operating Conditions Dialog Box
37.7.5. Select Input Parameter Dialog Box
37.7.6. Profiles Dialog Box
37.7.7. Orient Profile Dialog Box
37.7.8. Write Profile Dialog Box

37.7.1. Fluid Dialog Box

The **Fluid** dialog box sets the conditions for a fluid cell zone. It is opened from the Cell Zone Conditions Task Page (p. 2367). See Inputs for Fluid Zones (p. 253) and User Inputs for Porous Media (p. 267) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone Name

sets the name of the zone.

Material Name

sets the fluid material. The drop-down list contains the names of all materials that have been loaded into the solver. Materials are defined with the Materials Task Page (p. 2300).

Important

If you are modeling species transport or multiphase flow, the **Material Name** list will not appear in the **Fluid** dialog box. For species calculations, the mixture material for all fluid zones will be the material you specified in the Species Model Dialog Box (p. 2216). For

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

multiphase flows, the materials are specified when you define the phases, as described in Defining the Phases for the VOF Model (p. 1436).

Frame Motion
enables the moving reference frame model for the cell zone. See Specifying the Rotation Axis (p. 255) and Defining Zone Motion (p. 256) for details.

Mesh Motion
enables the sliding mesh model for the cell zone. See Setting Up the Sliding Mesh Problem (p. 634) for details.

Porous Zone
indicates that the zone is a porous medium. Additional items will appear in the dialog box when this option is enabled. See User Inputs for Porous Media (p. 267) for details.

3D Fan Zone
allows you to simulate the effect of an axial fan by applying a distributed momentum source in a toroid-shaped fluid volume. This model can calculate not only the axial velocity, but the swirl and radial velocities as well. When you enable this option, the **3D Fan Zone** tab becomes available, so that you can specify the geometry and properties of the fan. The cell zone should have a toroid shape that is sized to match the blade-swept volume of the fan you are simulating (that is, has an inner and outer radius that matches the radius of the fan's hub and blade tips, respectively, as well as a length that matches the thickness of the toroid region swept by the blades in the axial direction). Furthermore, the fluid cell zone should be "interior", that is, have at least two boundary zones of type **interior** that border another fluid cell zone and act as the fan inlet and outlet (the other boundaries can be of type **interior** as well). See 3D Fan Zones (p. 287) for details.

Laminar Zone
disables the calculation of turbulence production in the fluid zone (appears only for turbulent flow calculations using the Spalart-Allmaras model or one of the k- ε or k- ω models). See Specifying a Laminar Zone (p. 255) for details.

LES Zone
allows you to model a smaller embedded LES zone within a larger URANS computational domain for turbulent flow calculations. When you turn on this option, the **Embedded LES** tab will be enabled to allow you to specify properties for the embedded LES zone. See Setting Up the Embedded Large Eddy Simulation (ELES) Model (p. 793) for details.

Source Terms
enables the specification of volumetric sources of mass, momentum, energy, and so on. When you turn on this option, the **Source Terms** tab will be enabled to allow you to specify the values for the desired sources. See Defining Mass, Momentum, Energy, and Other Sources (p. 297) for details.

Fixed Values
enables the fixing of the value of one or more variables in the fluid zone, rather than computing them during the calculation. See Fixing the Values of Variables (p. 292) for details.

Important

You can fix values for velocity components, temperature, and species mass fractions only if you are using the pressure-based solver.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Participates In Radiation
specifies whether or not the fluid zone participates in radiation. This option appears when you are using the DO model for radiation.

> **Warning**
>
> In general, disabling **Participates in Radiation** for fluid zones is not recommended, as it can produce erroneous results. There are rare cases when it is acceptable: for example, if the domain contains multiple fluid zones, disabling this option for zones where radiation is negligible may save computational time without affecting the results.

Reaction
enables/disables reactions in the porous zone.

Electrolyte
indicates that the zone is an electrolyte zone. By default, all fluid zones are assumed to be electrolyte zones. This option appears when the **Electrochemical** Reaction model is enabled.

Reference Frame
lists the parameters that define motion for a moving reference frame.

Rotation-Axis Origin
specifies the origin for the axis of rotation of solid zone. See Specifying the Rotation Axis (p. 260) for details. This item will appear only for 3D and 2D non-axisymmetric models.

Rotation-Axis Direction
specifies the direction vector for the solid zone's axis of rotation. See Specifying the Rotation Axis (p. 260) for details. This item will appear only for 3D models.

Rotational Velocity
contains an input field for the rotational **Speed** of the zone.

Translational Velocity
contains inputs for the **X**, **Y**, and **Z** velocities of the zone.

Relative Specification
indicates whether the velocities are absolute velocities (**absolute**) or velocities relative to the motion of each cell zone (**Relative to Cell Zone**). This selection is necessary only if your problem involves moving reference frames or sliding meshes. If there is no zone motion, both options are equivalent.

UDF
allows you to hook the DEFINE_ZONE_MOTION user-defined function.

Copy to Mesh Motion
allows you to switch between the MRF and moving mesh models. The variables used for the origin, axis, and velocity components, as well as for the `DEFINE_ZONE_MOTION` user-defined function, will be copied from one model to the other.

Mesh Motion
lists the parameters that define motion for a moving reference frame.

Rotation-Axis Origin
specifies the origin for the axis of rotation of solid zone. See Specifying the Rotation Axis (p. 260) for details. This item will appear only for 3D and 2D non-axisymmetric models.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rotation-Axis Direction
specifies the direction vector for the solid zone's axis of rotation. See Specifying the Rotation Axis (p. 260) for details. This item will appear only for 3D models.

Rotational Velocity
contains an input field for the rotational **Speed** of the zone.

Translational Velocity
contains inputs for the **X**, **Y**, and **Z** velocities of the zone.

Relative Specification
indicates whether the velocities are absolute velocities (**absolute**) or velocities relative to the motion of each cell zone (**Relative to Cell Zone**). This selection is necessary only if your problem involves moving reference frames or sliding meshes. If there is no zone motion, both options are equivalent.

UDF
allows you to hook the DEFINE_ZONE_MOTION user-defined function.

Copy to Frame Motion
allows you to switch between the MRF and moving mesh models. The variables used for the origin, axis, and velocity components, as well as for the `DEFINE_ZONE_MOTION` user-defined function, will be copied from one model to the other.

Porous Zone
lists the parameters associated with the porous zone.

Conical
enables the specification of a conical (or cylindrical) porous medium. This item will appear only when the **Porous Zone** option is enabled for a 3D case.

Cone Half Angle
specifies the angle between the cone's axis and its surface (see Figure 6.13: Cone Half Angle (p. 271)). Set this to 0 to define the porous region using a cylindrical coordinate system. This item will appear only when the **Porous Zone** and **Conical** options are enabled.

Snap to Zone
aligns the plane (or line, in 2D) tool with the zone selected in the drop-down list. The tool is centered at the centroid of the zone, with the tool's axis normal to the zone. If this axis is not the desired cone axis, reposition the tool (as described in Using the Plane Tool (p. 1724)). When you are satisfied with the axis, click the **Update From Plane Tool** (or **Update From Line Tool**) button to update the **Cone Axis Vector** fields.

This item will appear only when the **Porous Zone** and **Conical** options are enabled.

Update From Plane Tool
(**Update From Line Tool** in 2D) updates the **Direction-1 Vector** and (in 3D) the **Direction-2 Vector** from the plane tool orientation. If the **Conical** option is enabled, this button will update the **Cone Axis Vector** and the **Point on Cone Axis**. See Defining the Viscous and Inertial Resistance Coefficients (p. 270) for details. This item will appear only when the **Porous Zone** option is enabled.

Direction-1 Vector, Direction-2 Vector
indicate the directions for which the resistance coefficients are defined. See Defining the Viscous and Inertial Resistance Coefficients (p. 270) for details. These items will appear only when the **Porous Zone** option is enabled, but the **Conical** option is not. (In 2D, only **Direction-1 Vector** will appear.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cone Axis Vector
specifies the **X,Y,Z** vector for the cone's axis.

This item will appear only when the **Porous Zone** and **Conical** options are enabled.

Point on Cone Axis
specifies a point on the cone's axis. This point will be used by ANSYS Fluent to transform the resistances to the Cartesian coordinate system.

This item will appear only when the **Porous Zone** and **Conical** options are enabled.

Relative Velocity Resistance Formulation
allows ANSYS Fluent to either apply the relative reference frame or the absolute reference frame. This allows for the correct prediction of the source terms.

Viscous Resistance, Inertial Resistance
contain inputs for the viscous resistance coefficient $1/\alpha$ and the inertial resistance coefficient $C2$ in each direction. See Defining the Viscous and Inertial Resistance Coefficients (p. 270) for details. These items will appear only when the **Porous Zone** option is enabled.

Alternative Formulation
provides better stability to the calculation when your porous medium is anisotropic.

If you have enabled the **Conical** option, **Direction-1** is the cone axis direction, **Direction-2** is the normal to the cone surface (radial (r) direction for a cylinder), and **Direction-3** is the circumferential (θ) direction.

Power Law Model
contains inputs for the **C0** and **C1** coefficients in the power law model for porous media. See Using the Power-Law Model (p. 276) for details.

Fluid Porosity
contains an additional input for the porous medium. See User Inputs for Porous Media (p. 267) for details.

Porosity
sets the volume fraction of fluid within the porous region.

Heat Transfer Settings
contains heat transfer settings for the porous medium. See Specifying the Heat Transfer Settings (p. 277) for details.

Thermal Model
specifies whether or not thermal equilibrium is assumed between the medium and the fluid flow.

Equilibrium
specifies that the medium and the fluid flow are in thermal equilibrium in the porous medium.

Non-Equilibrium
specifies that the medium and the fluid flow are not in thermal equilibrium in the porous medium, so that a dual cell approach is enabled.

Solid Material Name
specifies the solid material in the porous region. This drop-down menu is only available when **Equilibrium** is selected from the **Thermal Model** list.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solid Zone
displays the name of the solid cell zone that is coupled with the porous fluid zone through heat transfer. This field is only displayed when **Non-Equilibrium** is selected from the **Thermal Model** list.

Interfacial Area Density
specifies A_{fs} (as described in Non-Equilibrium Thermal Model Equations (p. 265)) for the porous region. This text-entry box is only available when **Non-Equilibrium** is selected from the **Thermal Model** list.

Heat Transfer Coefficient
specifies h_{fs} (as described in Non-Equilibrium Thermal Model Equations (p. 265)) for the porous region. This text-entry box is only available when **Non-Equilibrium** is selected from the **Thermal Model** list.

Anisotropic Species Diffusion
allows you to model anisotropic species diffusion in porous media. When this option is selected, you can specify the **Matrix Components** for the anisotropic diffusion matrix in the porous zone. This option is available only with the species transport models.

3D Fan Zone
provides the settings that define a 3D fan zone. See 3D Fan Zones (p. 287) for details.

Geometry
allows you to define the location and size of the fan.

Inlet Fan Zone
allows you to select a boundary zone of type **interior** that will act as the surface on which the fan will be located.

Hub Radius
defines the radius of the fan hub, that is, the inner radius of the blade-swept volume on which the fan's distributed momentum sources are applied.

Tip Radius
defines the radius of the blade tips of the fan, that is, the outer radius of the blade-swept volume on which the fan's distributed momentum sources are applied.

Thickness
defines the thickness of the toroid region swept by the blades in the axial direction.

Inflection Point
is a ratio that defines the fraction of the fan blade length (starting at the hub) over which the tangential velocity of the fan discharge is increasing with increasing radius and peaks, before tapering off (see Figure 6.17: The Inflection Point Ratio of a Pitched Blade Turbine (p. 291)).

Fan Origin
defines the **X**, **Y**, and **Z** coordinates of a point on the **Inlet Fan Zone** that is the origin for the fan.

Properties
allows you to define how the fan behaves.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rotational Direction
allows you to define the fan's rotational direction, using the right-hand rule with respect to the fan direction vector (that is, a vector from the **Fan Origin** that points into the 3D fan zone perpendicular to the **Inlet Fan Zone**).

Operating Angular Velocity
defines the angular velocity of the fan during the simulation.

Limit Flow Rate Through Fan
specifies that the flow rate through the fan adheres to minimum and maximum values, as defined in the **Flow Rate** group box.

Flow Rate
allows you to define the **Maximum** and **Minimum** flow rates through the fan. This group box is available when the **Limit Flow Rate Through Fan** option is enabled.

Tangential Source Term
enables a momentum source in the tangential direction, based on impeller theory.

Radial Source Term
enables a momentum source in the radial direction, based on impeller theory.

Axial Source Term
enables a momentum source in the axial direction, based on the settings in the **Axial Source Term Settings** group box.

Axial Source Term Settings
allows you to define the momentum source in the axial direction. This group box is only available when the **Axial Source Term** option is enabled.

Method
specifies whether a **constant pressure** or a **fan curve** (that is, a curve created from data points that plot pressure rise vs. volumetric flow rate) defines the axial momentum source.

Pressure Jump
defines the pressure jump that is applied across the 3D fan. This setting is only available when **constant pressure** is selected for the **Method**.

Read Fan Curve...
allows you to read a tab-delimited ASCII file that defines the relationship between pressure and flow rate for each 3D fan cell zone (see 3D Fan Curve File Format (p. 2869) for details). This button is only available when **fan curve** is selected for the **Method**.

Curve Fitting Method
specifies whether Fluent should apply a **polynomial** or **piecewise-linear** fitting method to construct a curve from the data points in the fan curve file. This drop-down list is only available when **fan curve** is selected for the **Method**.

Order of Polynomial
defines the order of the polynomial curve fitting method. This setting is only available when **polynomial** is selected for the **Curve Fitting Method**.

Initial Flow Rate
specifies your best estimate of the initial flow rate through the fan. This setting is only available when **fan curve** is selected for the **Method**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Test Angular Velocity
specifies the angular velocity of the fan used in the experiment that generated the fan curve data. This setting is only available when **fan curve** is selected for the **Method**.

Test Temperature
specifies the temperature of the fluid flowing through the fan during the experiment that generated the fan curve data. This setting is only available when **fan curve** is selected for the **Method**.

Reaction
lists the parameters for reactions in the porous zone.

Reaction Mechanism
allows you to specify a defined group, or mechanism, of available reactions. See Defining Zone-Based Reaction Mechanisms (p. 984) for details about defining reaction mechanisms.

Surface-to-Volume Ratio
specifies the surface area of the pore walls per unit volume ($\frac{A}{V}$), and can be thought of as a measure of catalyst loading. With this value, ANSYS Fluent can calculate the total surface area on which the reaction takes place in each cell by multiplying $\frac{A}{V}$ by the volume of the cell.

Source Terms
defines a source of heat, mass, momentum, turbulence, species, or other scalar quantity within the fluid zone.

Fixed Values
lists the fixed parameters of the fluid zone.

Local Coordinate System for Fixed Velocities
enables the specification of fixed cylindrical velocity components instead of Cartesian components. The local coordinate system is defined by the **Rotation-Axis Origin** and **Rotation-Axis Direction**.

This item is available only in 3D, and only when the **Fixed Values** option is on.

Multiphase
allows you to set parameters that are specific to the multiphase models.

Compressive Scheme Slope Limiter
ranges from 0 to 2. For example, a **Compressive Scheme Slope Limiter** of 0 corresponds to first order upwind behavior, a value of 1 corresponds to second order upwind, and a value of 2 applies the compressive scheme. The **Multiphase** tab is available only if **Zonal Discretization** is enabled in the Multiphase Model Dialog Box (p. 2171).

Numeric Beach Treatment
is available when **Open Channel Flow** and/or **Open Channel Wave BC** is enabled in the **Multiphase Model** dialog box (see Numerical Beach Treatment for Open Channels (p. 1432) for more detail).

Numerical Beach
when enabled expands the dialog box where you can specify the numerical beach parameters.

Beach Group ID
represents the cell zones sharing the damping length containing the same input parameters.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Damping Type
allows you to choose between **One Dimensional** and **Two Dimensional** damping.

One Dimensional
is the damping treatment in the flow direction.

Two Dimensional
is the damping treatment in the flow and gravity direction.

Compute From Inlet Boundary
is set to **none** by default. If there are available open channel boundaries (velocity-inlet, pressure-inlet, and mass-flow-inlet), boundary names are added to the drop-down list. If you select a boundary from the list, the **Level Inputs**, **Damping Length Inputs in Flow Direction**, and **Damping Resistance** values will be updated in the interface. You have the option to overwrite the updated inputs with values that are more applicable to your simulation.

Level Inputs
is only available for the **Two Dimensional** damping type.

Free Surface Level
is the same definition as for open channel flow, see Modeling Open Channel Flows (p. 1410).

Bottom Level
is the same definition as for open channel flow, see Modeling Open Channel Flows (p. 1410), and is valid only for shallow waves. The bottom level is used for calculating the liquid height.

Flow Direction
is the X, Y, and Z (for 3D) components.

Damping Length Inputs in Flow Direction
are required to calculate the start and end points of the damping length in the flow direction.

Damping Length Specification
is only available if **Open Channel Wave BC** is enabled in the **Multiphase Model** dialog box. There are two options you can choose from **End Point and Wave Lengths** or **End and Start Points**.

End Point
is the end point of the damping zone.

Start Point
is the starting point in the flow direction.

Wave Length
is updated automatically if the boundary is selected from the **Compute From Inlet Boundary** drop-down list.

Number Of Wave Lengths
is set to 2 by default for the calculation of the damping length.

Relative Velocity Resistance Formulation
calculates the source term using relative velocities in the numerical beach zone when using moving/deforming meshes or moving reference frames.

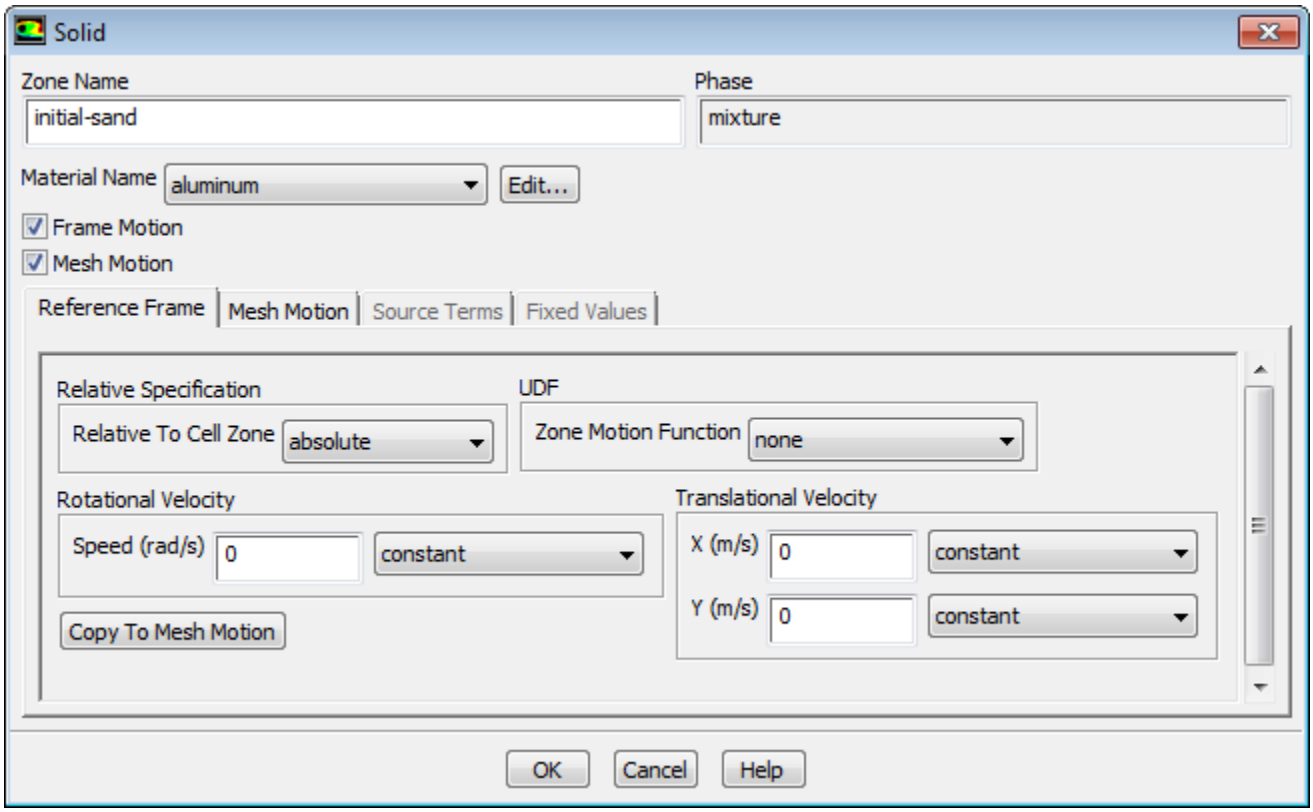
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Linear Damping Resistance
is the resistance per unit time.

Quadratic Damping Resistance
is the resistance per unit length.

37.7.2. Solid Dialog Box

The **Solid** dialog box sets the boundary conditions for a solid cell zone. It is opened from the Cell Zone Conditions Task Page (p. 2367). See Inputs for Solid Zones (p. 259) for details about the items below.



Controls

Zone Name
sets the name of the zone.

Material Name
selects the material type of the solid. Materials are defined with the Materials Task Page (p. 2300).

Frame Motion
enables the moving reference frame model for the cell zone. See Specifying the Rotation Axis (p. 255) and Defining Zone Motion (p. 256) for details.

Mesh Motion
enables the sliding mesh model for the cell zone. See Setting Up the Sliding Mesh Problem (p. 634) for details.

Source Terms
enables the specification of a volumetric source of energy. When you turn on this option, the **Source Term** tab will allow you to specify the value for the energy source. See Defining Mass, Momentum, Energy, and Other Sources (p. 297) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fixed Values
enables the fixing of the value of temperature in the solid zone, rather than computing it during the calculation. See Fixing the Values of Variables (p. 292) for details.

> **Important**
>
> You can fix the value of temperature only if you are using the pressure-based solver.

Participates In Radiation
specifies whether or not the solid zone participates in radiation. This option appears when you are using the DO model for radiation.

Reference Frame
lists the parameters that define motion for a moving reference frame.

Rotation-Axis Origin
specifies the origin for the axis of rotation of solid zone. See Specifying the Rotation Axis (p. 260) for details. This item will appear only for 3D and 2D non-axisymmetric models.

Rotation-Axis Direction
specifies the direction vector for the solid zone's axis of rotation. See Specifying the Rotation Axis (p. 260) for details. This item will appear only for 3D models.

Rotational Velocity
contains an input field for the rotational **Speed** of the zone.

Translational Velocity
contains inputs for the **X**, **Y**, and **Z** velocities of the zone.

Relative Specification
indicates whether the velocities are absolute velocities (**absolute**) or velocities relative to the motion of each cell zone (**Relative to Cell Zone**). This selection is necessary only if your problem involves moving reference frames or sliding meshes. If there is no zone motion, both options are equivalent.

UDF
allows you to hook the DEFINE_ZONE_MOTION user-defined function.

Copy to Mesh Motion
allows you to switch between the MRF and moving mesh models. The variables used for the origin, axis, and velocity components, as well as for the `DEFINE_ZONE_MOTION` user-defined function, will be copied from one model to the other.

Mesh Motion
lists the parameters that define motion for a moving reference frame.

Rotation-Axis Origin
specifies the origin for the axis of rotation of solid zone. See Specifying the Rotation Axis (p. 260) for details. This item will appear only for 3D and 2D non-axisymmetric models.

Rotation-Axis Direction
specifies the direction vector for the solid zone's axis of rotation. See Specifying the Rotation Axis (p. 260) for details. This item will appear only for 3D models.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rotational Velocity
contains an input field for the rotational **Speed** of the zone.

Translational Velocity
contains inputs for the **X**, **Y**, and **Z** velocities of the zone.

Relative Specification
indicates whether the velocities are absolute velocities (**absolute**) or velocities relative to the motion of each cell zone (**Relative to Cell Zone**). This selection is necessary only if your problem involves moving reference frames or sliding meshes. If there is no zone motion, both options are equivalent.

UDF
allows you to hook the DEFINE_ZONE_MOTION user-defined function.

Copy to Frame Motion
allows you to switch between the MRF and moving mesh models. The variables used for the origin, axis, and velocity components, as well as for the `DEFINE_ZONE_MOTION` user-defined function, will be copied from one model to the other.

Source Term
lists the parameters for volumetric source of energy.

Energy
displays the total number of energy sources used.

User Scalar n
displays the total number of scalars used.

Fixed Values
lists the parameters that can be declared as fixed during the calculation.

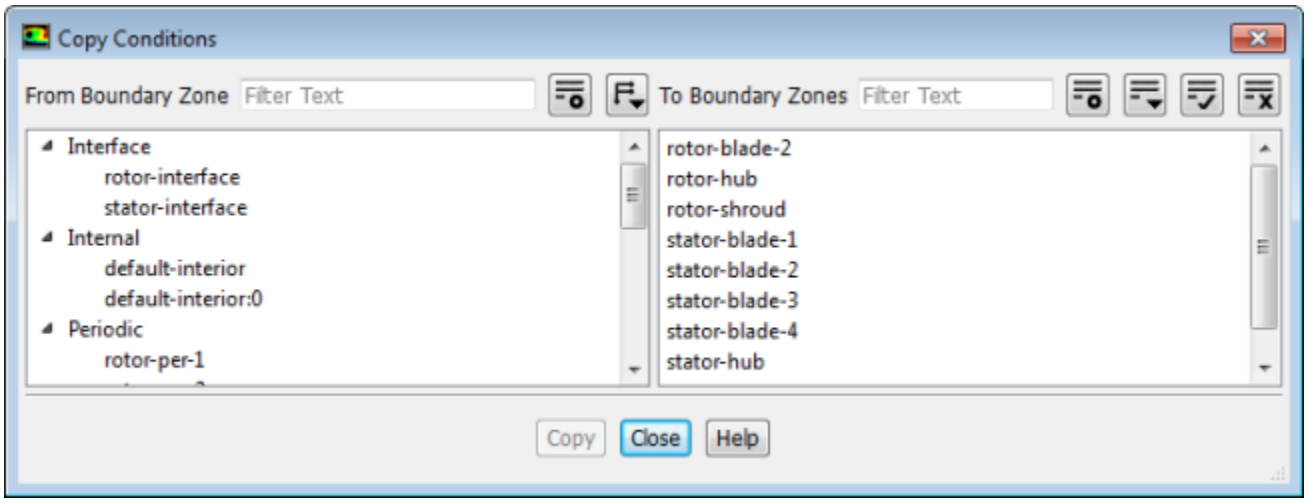
Temperature
specifies the fixed value for temperature.

User Scalar n
specifies the fixed value for user scalar.

37.7.3. Copy Conditions Dialog Box

The **Copy Conditions** dialog box allows you to copy cell zone and/or boundary conditions from one zone/boundary to other zones/boundaries of the same type. It is opened either from the Cell Zone Conditions Task Page (p. 2367) or from the Boundary Conditions Task Page (p. 2389). See Copying Cell Zone and Boundary Conditions (p. 242) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

From Zone
specifies the zone that has the conditions you want to copy.

To Zones
specifies the zone or zones to which you want to copy the conditions.

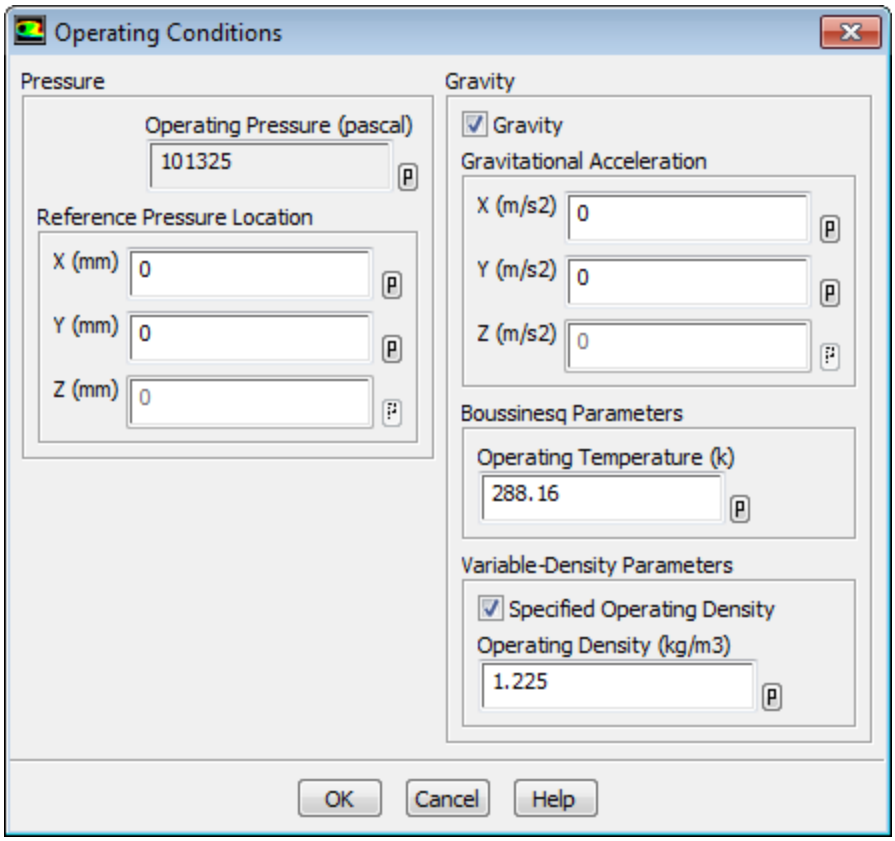
Phase
specifies the phase for which cell zone conditions or boundary conditions are being copied. This item appears if the VOF, mixture, or Eulerian multiphase model is being used. See Steps for Copying Cell Zone and Boundary Conditions (p. 1406) for details.

Copy
copies the cell zone conditions or boundary conditions, setting all of the conditions for the zones selected in the **To Zones** list to be the same as the conditions for the zone selected in the **From Zone** list.

37.7.4. Operating Conditions Dialog Box

The **Operating Conditions** dialog box allows you to set parameters related to operating conditions in your model.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Pressure

contains items related to the modeling of pressure.

Floating Operating Pressure

specifies the use of a floating operating pressure. See Floating Operating Pressure (p. 597) for details. This option appears only for time-dependent compressible flows.

Operating Pressure

sets the operating pressure for the problem. For all flows, ANSYS Fluent uses gauge pressure internally. Any time an absolute pressure is needed, it is generated by adding the operating pressure to the relative pressure. See Operating Pressure (p. 530) for a detailed description of operating pressure and how to set it.

Reference Pressure Location

sets the location of the cell whose pressure value is used to adjust the gauge pressure field for incompressible flows that do not involve any pressure boundaries. See Reference Pressure Location (p. 531) for details.

Real Gas Phase

allows you to specify the operating conditions in the subcritical regime of your model. Note that if the operating conditions in your model are entirely in the supercritical regime, this setting will have no effect.

Note

This option appears only when a Cubic Equation of State Real Gas model is chosen.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Vapor
is the default option, and therefore assumes that the real gas phase is vapor.

Liquid
assumes that the real gas phase is liquid.

Gravity
contains inputs for gravitational acceleration, the Boussinesq model, and variable density.

Gravity
enables the specification of gravity.

Gravitational Acceleration
sets the x, y, and z components of the gravitational acceleration vector. (The z component is available only in 3D solvers.) See Natural Convection and Buoyancy-Driven Flows (p. 833) for details about buoyancy-driven flows. This option appears only when **Gravity** is enabled.

Boussinesq Parameters
contains inputs related to the Boussinesq model. This option appears only if **Energy** (in the Energy Dialog Box (p. 2175)) and **Gravity** are enabled. See The Boussinesq Model (p. 833) for more information on the Boussinesq model.

Operating Temperature
sets the operating temperature (T_0 in Equation 12.3 (p. 833)) for use with the Boussinesq approximation.

Variable-Density Parameters
contains inputs related to the modeling of variable density. This option appears only when **Gravity** is enabled.

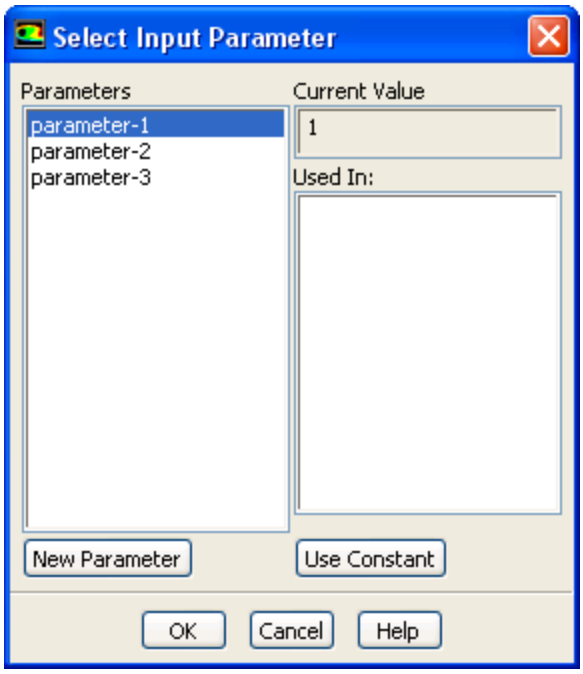
Specified Operating Density
enables the specification of operating density. See Operating Density (p. 836) for details.

Operating Density
sets the operating density (ρ_0 in Equation 12.4 (p. 836)). This parameter can be set only when **Specified Operating Density** is enabled.

37.7.5. Select Input Parameter Dialog Box

The **Select Input Parameter** dialog box allows you to choose from a listing of existing input parameters as well as to create and define new input parameters. For more information about parameters, see Defining and Viewing Parameters (p. 243) in the User's Guide, and see Working With Input and Output Parameters in Workbench in the ANSYS Fluent in Workbench User's Guide.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Parameters
contains a list of existing compatible input parameters

Current Value
displays the value of the currently selected parameter.

Used In
lists any variables that are already associated with the currently selected parameter.

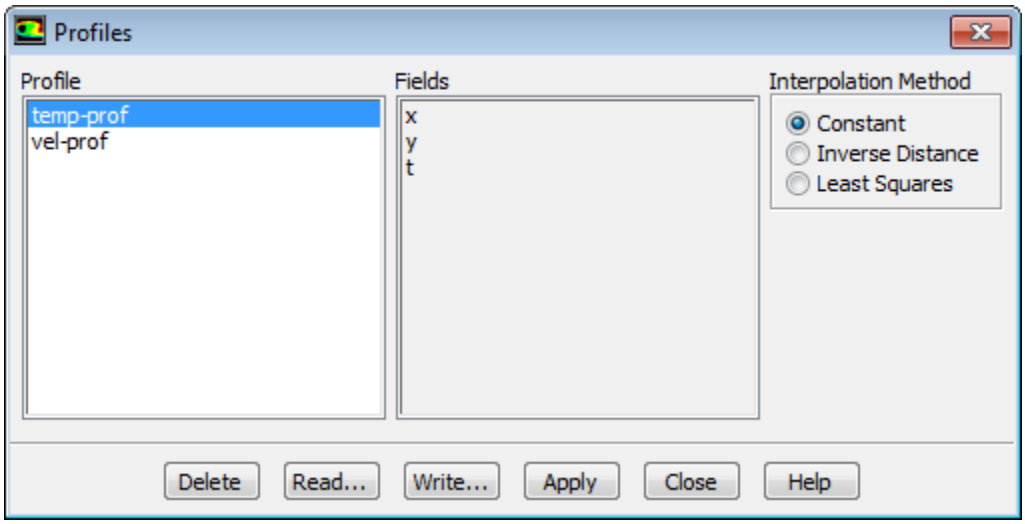
Use Constant
allows you to change the associated parameter to a constant (that is, real) value.

New Parameter
opens the Input Parameter Properties Dialog Box (p. 2631), in which you can assign names and values to an input parameter.

37.7.6. Profiles Dialog Box

The **Profiles** dialog box allows you to define new profiles by reading cell zone and boundary profile files. You can also examine the existing profile definitions and delete unused profiles. See Profiles (p. 435) for details about cell zone and boundary profiles.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Profile

contains a selectable list of available profiles. When a profile is selected its available fields are displayed under **Fields**.

Fields

displays the fields available in the selected profile. After the profile file has been read, these fields will also appear in any boundary condition dialog box (for example, the **Velocity Inlet** dialog box) that allows profile specification of a variable. To the right of (or below) the variable in the boundary conditions dialog box, there will be a drop-down list that contains a **constant** and the fields from available profile files. To use a particular profile field, just select it from the list.

Interpolation Method

allows you to select the interpolation method for the profile selected from the **Profile** list. This selection is only available for point profiles, and will only take effect when the **Apply** button is clicked. The choices include the following:

Constant

specifies that ANSYS Fluent should use zeroth-order interpolation to assign the point profile values to the nearest cell faces at the boundary. This is the default selection.

Inverse Distance

specifies that ANSYS Fluent should assign a value to each cell face at the boundary based on weighted contributions from the values in the point profile file. The weighting factor is inversely proportional to the distance between the profile point and the cell face center.

Least Squares

specifies that ANSYS Fluent should assign values to the cell faces at the boundary through a first-order interpolation method that tries to minimizes the sum of the squares of the offsets (residuals) between the profile data points and the cell face centers.

Delete

deletes the selected profile from memory.

Orient...

opens the Orient Profile Dialog Box (p. 2387), in which you can reorient and scale the profile. This item appears only in 3D.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

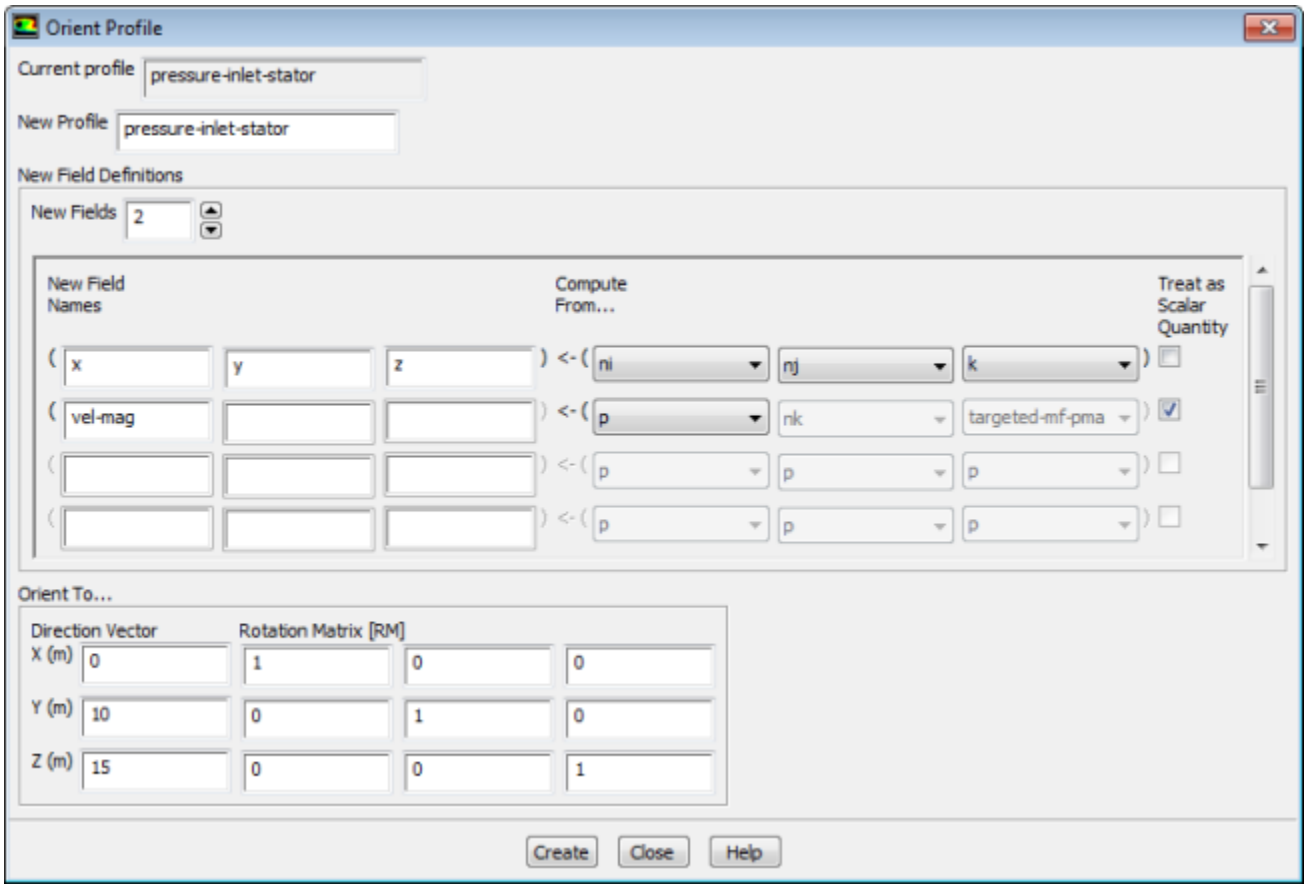
Read...
opens The Select File Dialog Box (p. 19) so that you can read a boundary profile file. If a profile in the file has the same name as an existing profile, the old profile will be overwritten.

Write...
opens the Write Profile Dialog Box (p. 2388), in which you can save profile data.

Apply
sets the selection made in the **Interpolation Method** list for point profiles in preparation for interpolation. The profile is not actually interpolated until a profile field is selected in a boundary condition dialog box (for example, the **Velocity Inlet** dialog box) and the solution is initialized.

37.7.7. Orient Profile Dialog Box

The **Orient Profile** dialog box allows you to reorient a profile so that you can apply it to a particular boundary. See Reorienting Profiles (p. 443) for details.



Controls

Current profile
shows the name of the currently selected profile in the Profiles Dialog Box (p. 2385). This is the profile on which the new profile will be based.

New Profile
sets the name of the new cell zone or boundary profile.

New Field Definitions
contains inputs and controls for the definition of the vector and scalar fields in the new profile.

New Fields
sets the number of data fields in the new profile.

New Field Names
contains inputs for the names of the data fields in the new profile. For a vector field, all 3 inputs in each row will be active; for a scalar field, only the first will be active.

Compute From...
contains drop-down lists with the names of the fields in the **Current profile**. In these lists, select the fields from which the **New Field Names** will be computed.

Treat as Scalar Quantity
indicates (if on) that the adjacent field is a scalar quantity. If this option is off, the field is a vector quantity.

Orient To...
contains inputs for the definition of the local coordinate system for the new profile. This coordinate system will determine the orientation of the profile.

Direction Vector
is the vector that translates a cell zone or boundary profile to the new position, and is defined between the centers of the profile fields.

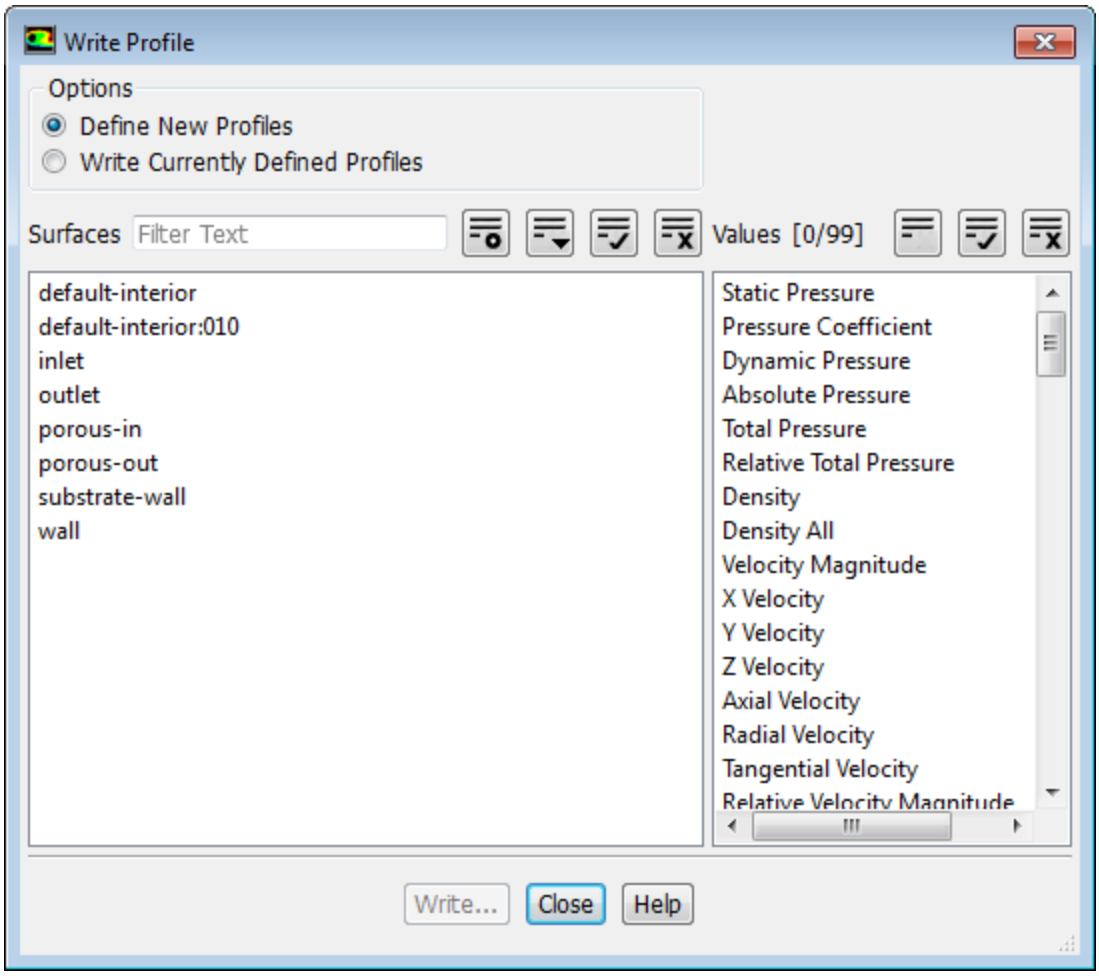
Rotation Matrix [RM
] specifies the rotational matrix RM, which is based on Euler angles (γ, β, and α) that define an orthogonal system $x'y'z'$ as the result of the three successive rotations from the original system xyz. See Reorienting Profiles (p. 443) for more information.

Create
creates a new profile using the information specified in the dialog box.

37.7.8. Write Profile Dialog Box

The **Write Profile** dialog box allows you to create a profile file from the conditions on a specified cell zone or boundary/surface. See Writing Profile Files (p. 54) for details on writing profile files.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options
contains options for writing profiles.

Define New Profiles
enables the creation of a profile file from the conditions on a specified boundary or surface.

Write Currently Defined Profiles
enables the creation of a profile file containing all profiles that are currently defined.

Surfaces
contains a list from which you can select the surface(s) from which you want to extract data.

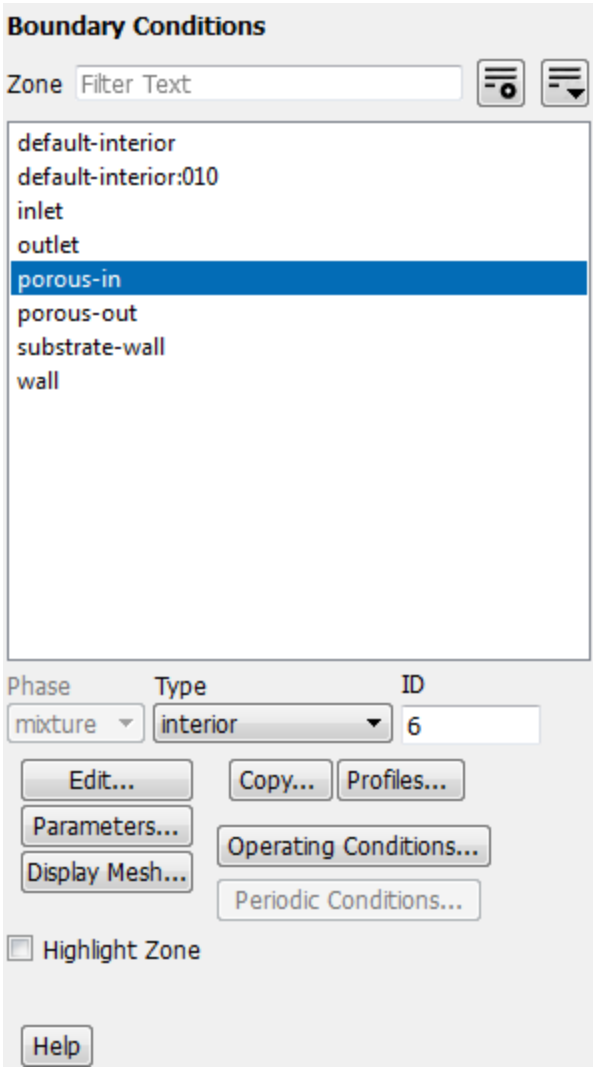
Values
contains a list from which you can select the variable(s) for which you want to create profiles.

Write...
opens The Select File Dialog Box (p. 19), in which you can specify a filename for the profile file.

37.8. Boundary Conditions Task Page

The **Boundary Conditions** task page allows you to set the type of a boundary and display other dialog boxes to set the boundary condition parameters for each boundary.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone

contains a selectable list of boundary zones from which you can select the zone of interest. You can check a zone type by using the mouse probe (see Controlling the Mouse Button Functions (p. 1800)) on the displayed physical mesh. This feature is particularly handy if you are setting up a problem for the first time, or if you have two or more boundary zones of the same type and you want to determine the zone IDs. To do this you must first display the mesh with the Mesh Display Dialog Box (p. 2163). Then click the boundary zone with the right (select) mouse button. ANSYS Fluent will print the zone ID and type of that boundary zone in the console window.

Phase

specifies the phase for which conditions at the selected boundary **Zone** are being set. This item appears if the VOF, mixture, or Eulerian multiphase model is being used. See Defining Multiphase Cell Zone and Boundary Conditions (p. 1391) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type
contains a drop-down list of boundary condition types for the selected zone. The list contains all possible types to which the zone can be changed.

Important

Note that you cannot use this method to change zone types to or from the periodic type, since additional restrictions exist for this boundary type. Creating Conformal Periodic Zones (p. 214) explains how to create and uncouple periodic zones.

ID
displays the zone ID number of the selected zone. (This is for informational purposes only; you cannot edit this number.)

Edit...
opens the appropriate dialog box for setting the boundary conditions for that particular boundary type.

Copy...
opens the Copy Conditions Dialog Box (p. 2381), which allows you to copy boundary conditions from one zone to other zones of the same type. See Copying Cell Zone and Boundary Conditions (p. 242) for details.

Profiles...
opens the Profiles Dialog Box (p. 2385).

Parameters...
opens the Parameters Dialog Box (p. 2627).

Operating Conditions...
opens the Operating Conditions Dialog Box (p. 2382).

Display Mesh...
opens the Mesh Display Dialog Box (p. 2163).

Periodic Conditions...
opens the Periodic Conditions Dialog Box (p. 2468).

Highlight Zone
when enabled highlights the boundary zone (selected in the task page) in the graphics window.

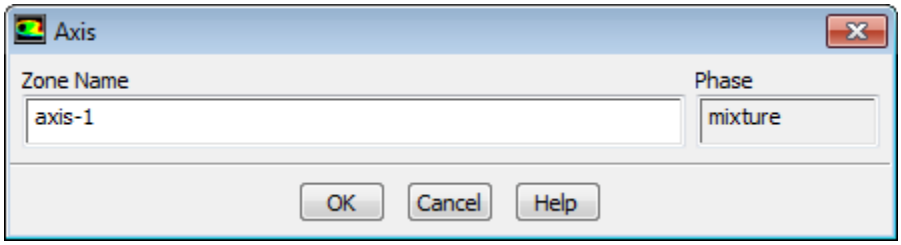
For additional information, see the following sections:
37.8.1. Axis Dialog Box
37.8.2. Degassing Dialog Box
37.8.3. Exhaust Fan Dialog Box
37.8.4. Fan Dialog Box
37.8.5. Inlet Vent Dialog Box
37.8.6. Intake Fan Dialog Box
37.8.7. Interface Dialog Box
37.8.8. Interior Dialog Box
37.8.9. Mass-Flow Inlet Dialog Box
37.8.10. Mass Flow Outlet Dialog Box
37.8.11. Outflow Dialog Box
37.8.12. Outlet Vent Dialog Box
37.8.13. Periodic Dialog Box

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.8.14. Porous Jump Dialog Box
37.8.15. Pressure Far-Field Dialog Box
37.8.16. Pressure Inlet Dialog Box
37.8.17. Pressure Outlet Dialog Box
37.8.18. Radiator Dialog Box
37.8.19. RANS/LES Interface Dialog Box
37.8.20. Symmetry Dialog Box
37.8.21. Velocity Inlet Dialog Box
37.8.22. Wall Dialog Box
37.8.23. Periodic Conditions Dialog Box

37.8.1. Axis Dialog Box

The **Axis** dialog box can be used to modify the name of an axis zone; there are no conditions to be set. It is opened from the Boundary Conditions Task Page (p. 2389). See Axis Boundary Conditions (p. 389) for information about axis boundaries.



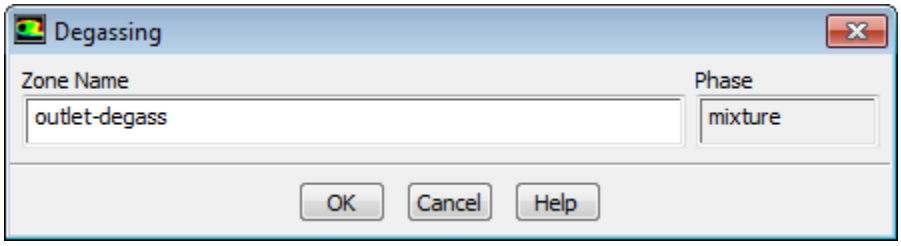
Controls

Zone Name
sets the name of the zone.

Phase
displays the name of the phase. This item appears if the VOF, mixture, or Eulerian multiphase model is being used.

37.8.2. Degassing Dialog Box

The **Degassing** dialog box can be used to modify the name of a degassing zone; there are no conditions to be set. It is opened from the Boundary Conditions Task Page (p. 2389). See Degassing Boundary Conditions (p. 357) for information about axis boundaries.



Controls

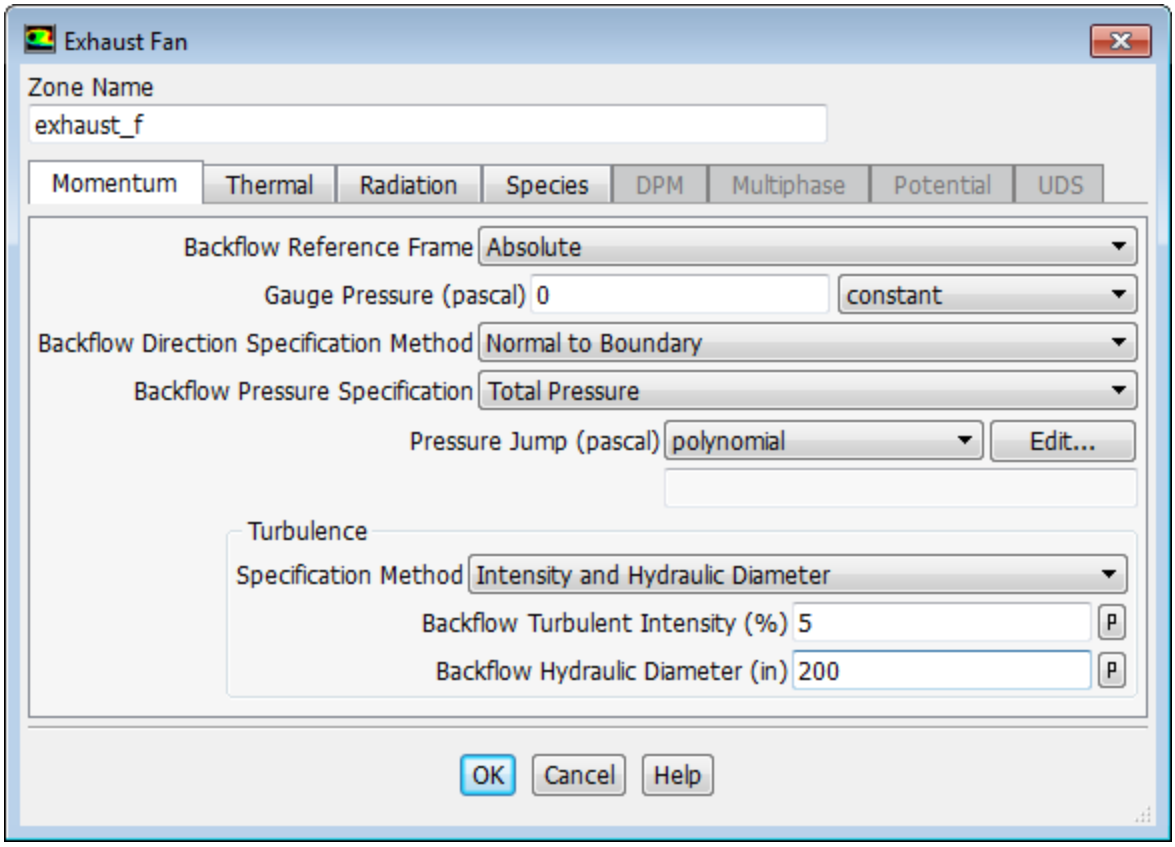
Zone Name
sets the name of the zone.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Phase
displays the name of the phase.

37.8.3. Exhaust Fan Dialog Box

The **Exhaust Fan** dialog box sets the boundary conditions for an exhaust fan zone. It is opened from the Boundary Conditions Task Page (p. 2389). See Inputs at Exhaust Fan Boundaries (p. 355) for details about defining the items below.



Controls

Zone Name
sets the name of the zone.

Phase
displays the name of the phase. It appears only for multiphase flows.

Momentum
contains the momentum parameters.

Backflow Reference Frame
specify whether backflow temperature, pressure, and flow directions are in the **Absolute** or **Relative to the Adjacent Cell Zone** reference frame.

Gauge Pressure
sets the gauge pressure at the outlet boundary.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Backflow Direction Specification Method
sets the direction of the inflow stream should the flow reverse direction. You can choose **Direction Vector**, **Normal to Boundary**,or **From Neighboring Cell**.

Coordinate System
contains a drop-down list for selecting the coordinate system. You can choose **Cartesian**, **Cylindrical**, or **Local Cylindrical**. This option is available only when **Direction Vector** is selected from the **Backflow Direction Specification Method** drop-down list.

X-, Y-, Z-Component of Flow Direction
allows you to specify the velocity components in x, y, and z directions respectively. This option is available when **Cartesian** is selected for the **Coordinate System**.

Radial-, Tangential-, Axial-Component of Flow Direction
set the direction of the flow at the boundary. These items will appear for 2D axisymmetric cases, or for 3D cases for which the selected **Coordinate System** is **Cylindrical** or **Local Cylindrical**.

Backflow Pressure Specification
specifies how the pressure is calculated under backflow conditions. If you select **Static Pressure**, the **Gauge Pressure** is directly imposed as the boundary face pressure; if you select **Total Pressure**, the **Gauge Pressure** will be combined with a dynamic contribution that is based on the velocity in the adjacent cell zone.

Pressure Jump
specifies the rise in pressure across the fan. See Specifying the Pressure Jump (p. 356) for details.

Axis Origin
sets the **X**, **Y**, and **Z** coordinates of the origin of the local cylindrical coordinate system.

Axis Direction
sets the **X**, **Y**, and **Z** components of the direction of the local cylindrical coordinate system.

Turbulence
displays the turbulence parameters.

Specification Method
specifies which method will be used to define the turbulence parameters. You can choose **K and Epsilon** (k- ε models and RSM only), **K and Omega** (k- ω models only), **Intensity and Length Scale**, **Intensity and Viscosity Ratio**, **Intensity and Hydraulic Diameter**, **Modified Turbulent Viscosity** (Spalart-Allmaras model only), or **Turbulent Viscosity Ratio** (Spalart-Allmaras model only). See Determining Turbulence Parameters (p. 303) for information about the inputs for each of these methods. (This item will appear only for turbulent flow calculations.)

Backflow Turbulent Kinetic Energy, Backflow Turbulent Dissipation Rate
set values for the turbulence kinetic energy k and its dissipation rate ε. These items will appear if you choose **K and Epsilon** as the **Specification Method**.

Backflow Turbulent Kinetic Energy, Backflow Specification Dissipation Rate
set values for the turbulence kinetic energy k and its specific dissipation rate ω. These items will appear if you choose **K and Omega** as the **Specification Method**.

Backflow Turbulent Intensity, Backflow Turbulent Length Scale
set values for turbulence intensity I and turbulence length scale ℓ. These items will appear if you choose **Intensity and Length Scale** as the **Specification Method**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Backflow Turbulent Intensity, Backflow Turbulent Viscosity Ratio

set values for turbulence intensity I and turbulent viscosity ratio μ_t / μ. These items will appear if you choose **Intensity and Viscosity Ratio** as the **Specification Method**.

Backflow Turbulent Intensity, Backflow Hydraulic Diameter

set values for turbulence intensity I and hydraulic diameter L. These items will appear if you choose **Intensity and Hydraulic Diameter** as the **Specification Method**.

Backflow Modified Turbulent Viscosity

sets the value of the backflow modified turbulent viscosity $\tilde{\nu}$. This item will appear if you choose **Modified Turbulent Viscosity** as the **Specification Method**.

Backflow Turbulent Viscosity Ratio

sets the value of the backflow turbulent viscosity ratio μ_t / μ. This item will appear if you choose **Turbulent Viscosity Ratio** as the **Specification Method**.

Reynolds-Stress Specification Method

specifies which method will be used to determine the backflow Reynolds stress boundary conditions when the Reynolds stress turbulence model is used. You can choose either **K or Turbulence Intensity** or **Reynolds-Stress Components**. If you choose the former, ANSYS Fluent will compute the Reynolds stresses for you. If you choose the latter, you will explicitly specify the Reynolds stresses yourself. See Reynolds Stress Model (p. 808) for details. (This item will appear only for RSM turbulent flow calculations.)

Backflow UU, VV, WW, UV, VW, UW Reynolds Stresses

specify the backflow Reynolds stress components when **Reynolds-Stress Components** is chosen as the **Reynolds-Stress Specification Method**.

Thermal

contains the thermal parameters. This parameter is available only when the energy equation is turned on.

Backflow Total Temperature

sets the total temperature of the inflow stream should the flow reverse direction.

Radiation

contains the boundary conditions for the radiation model at the exhaust fan.

External Black Body Temperature Method, Internal Emissivity

set the radiation boundary conditions when you are using the P-1, DTRM, DO, S2S, or MC models for radiation heat transfer. See Defining Boundary Conditions for Radiation (p. 869) for details.

Participates in Solar Ray Tracing

specifies whether or not the fan participates in solar ray tracing.

Solar Transmissivity Factor

specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the fan.

Participates in View Factor Calculation

specifies whether or not the fan participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Species
contains the species parameters.

Specify Species in Mole Fractions
allows you to specify the species in mole fractions rather than mass fractions.

Mean Mixture Fraction, Mixture Fraction Variance
set inlet values for the PDF mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model.)

Secondary Mean Mixture Fraction, Secondary Mixture Fraction Variance
set inlet values for the secondary mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model with two mixture fractions.)

Species Mass Fractions
contains inputs for the mass fractions of defined species. See Defining Cell Zone and Boundary Conditions for Species (p. 990) for details about these inputs. These items will appear only if you are modeling non-reacting multi-species flow or you are using the finite-rate reaction formulation.

Backflow Progress Variable
sets the value of the progress variable for premixed turbulent combustion. See Setting Boundary Conditions for the Progress Variable (p. 1094) for details.

This item will appear only if the premixed or partially premixed combustion model is used.

DPM
contains the discrete phase parameters. This tab is available only if you have defined at least one injection.

Discrete Phase BC Type
sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect
rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution (see Particle Reflection at Wall in the *Fluent Theory Guide*).

trap
terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 23.25: "Trap" Boundary Condition for the Discrete Phase (p. 1303).

escape
reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 23.26: "Escape" Boundary Condition for the Discrete Phase (p. 1303).

wall-jet
indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 16.6: "Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

user-defined
specifies a user-defined function to define the discrete phase boundary condition type.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discrete Phase BC Function
sets the user-defined function from the drop-down list.

Multiphase
contains the multiphase parameters.

Backflow Granular Temperature
specifies temperature for the solids phase and is proportional to the kinetic energy of the random motion of the particles.

Volume Fraction Specification Method
sets the method used to specify the volume fraction of the secondary phase selected in the Boundary Conditions Task Page (p. 2389). This section of the dialog box will appear when one of the multiphase models is being used. See Defining Multiphase Cell Zone and Boundary Conditions (p. 1391) for details.

Backflow Volume Fraction
specifies the volume fraction of the secondary phase as a constant, profile, of UDF function.

From Neighboring Cell
calculates the volume fraction from the neighboring cells.

Potential
displays the boundary conditions for the electric potential field. This tab is available only if you have enabled either the **Electric Potential model** or the **Electrochemical** reaction model in the Species Dialog Box (p. 2332).

Potential Boundary Condition
is a drop-down list of available potential boundary condition types: **Specified Flux** and **Specified Value**. For the **Specified Flux** boundary condition, you will need to specify **Current Density** at the wall. For the **Specified Value** boundary condition, you will need to specify **Potential** at the wall.

UDS
contains the UDS parameters.

User-Defined Scalar Boundary Condition
appears only if user-defined scalars are specified.

User Scalar n
specifies whether the scalar is a specified flux or a specified value.

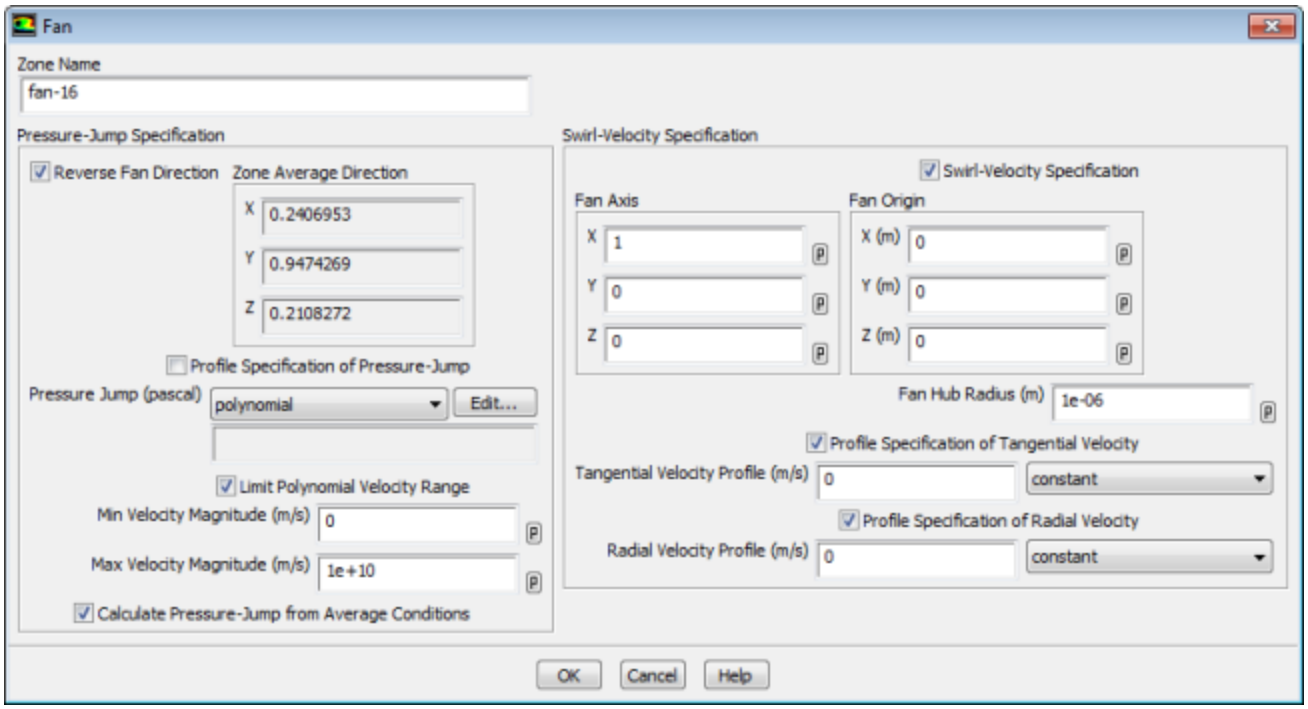
User-Defined Scalar Boundary Value
appears only if user-defined scalars are specified.

User Scalar n
specifies the value of the scalar.

37.8.4. Fan Dialog Box

The **Fan** dialog box sets the boundary conditions for a fan zone. It is opened from the Boundary Conditions Task Page (p. 2389). See User Inputs for Fans (p. 391) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone Name
sets the name of the zone.

Pressure-Jump Specification
contains inputs that define the pressure jump across the fan.

Reverse Fan Direction
sets the fan flow direction relative to the zone direction. If **Zone Average Direction** is pointing in the direction you want the fan to blow, do *not* select **Reverse Flow**; if it is pointing in the opposite direction, select **Reverse Flow**.

Zone Average Direction
displays the (face-averaged) direction vector for the zone as an aid in determining whether or not you want to select **Reverse Flow**.

Profile Specification of Pressure-Jump
enables the use of a boundary profile or user-defined function for the pressure jump specification. See Profiles (p. 435) or the Fluent Customization Manual for details. When this option is enabled, **Pressure Jump Profile** will appear in the dialog box and the next four items below it will not.

Pressure Jump Profile
contains a drop-down list from which you can select a boundary profile or a user-defined function for the pressure jump definition. This item will appear if you enable **Profile Specification of Pressure-Jump**.

Pressure-Jump
specifies the pressure-jump as a constant value or as a polynomial, piecewise-linear, or piecewise-polynomial function of velocity. See Defining the Pressure Jump (p. 392) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Limit Polynomial Velocity Range
limits the minimum and maximum velocity magnitudes used to calculate the pressure jump when it is defined as a function of velocity.

Min Velocity Magnitude, Max Velocity Magnitude
specify the minimum and maximum values to which the velocity magnitude is limited (when the **Limit Polynomial Velocity Range** option is enabled).

Calculate Pressure-Jump from Average Conditions
enables the option to use the mass-averaged velocity normal to the fan to determine a single pressure-jump value for all faces in the fan zone.

Discrete Phase BC Type
sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

interior
allows the particles to pass through the boundary.

reflect
rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See Particle Reflection at Wall in the *Fluent Theory Guide*.)

trap
terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 23.25: "Trap" Boundary Condition for the Discrete Phase (p. 1303).

escape
reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 23.26: "Escape" Boundary Condition for the Discrete Phase (p. 1303).

wall-jet
indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 16.6: "Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

user-defined
specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function
sets the user-defined function from the drop-down list.

Swirl-Velocity Specification
contains inputs for the specification of fan swirl velocity. This section of the dialog box appears only for 3D models.

Swirl-Velocity Specification
enables the specification of a swirl velocity for the fan.

Fan Axis
sets the direction vector for the fan's axis of rotation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fan Origin
sets the origin in the global coordinate system through which the fan rotation axis passes.

Fan Hub Radius
set the radius of the hub. The default is 1e-6 to avoid division by zero in the polynomial.

Profile Specification of Tangential Velocity
enables the use of a boundary profile or user-defined function for the tangential velocity specification. See Profiles (p. 435) or the Fluent Customization Manual for details. When this option is enabled, **Tangential Velocity Profile** will appear in the dialog box and **Tangential-Velocity Polynomial Coefficients** will not.

Tangential Velocity Profile
contains a drop-down list from which you can select a boundary profile or a user-defined function for the definition of the tangential velocity. This item will appear if you enable **Profile Specification of Tangential Velocity**.

Tangential-Velocity Polynomial Coefficients
sets the coefficients for the tangential velocity polynomial. Separate the coefficients by spaces.

Profile Specification of Radial Velocity
enables the use of a boundary profile or user-defined function for the radial velocity specification. See Profiles (p. 435) or the Fluent Customization Manual for details. When this option is enabled, **Radial Velocity Profile** will appear in the dialog box and **Radial-Velocity Polynomial Coefficients** will not.

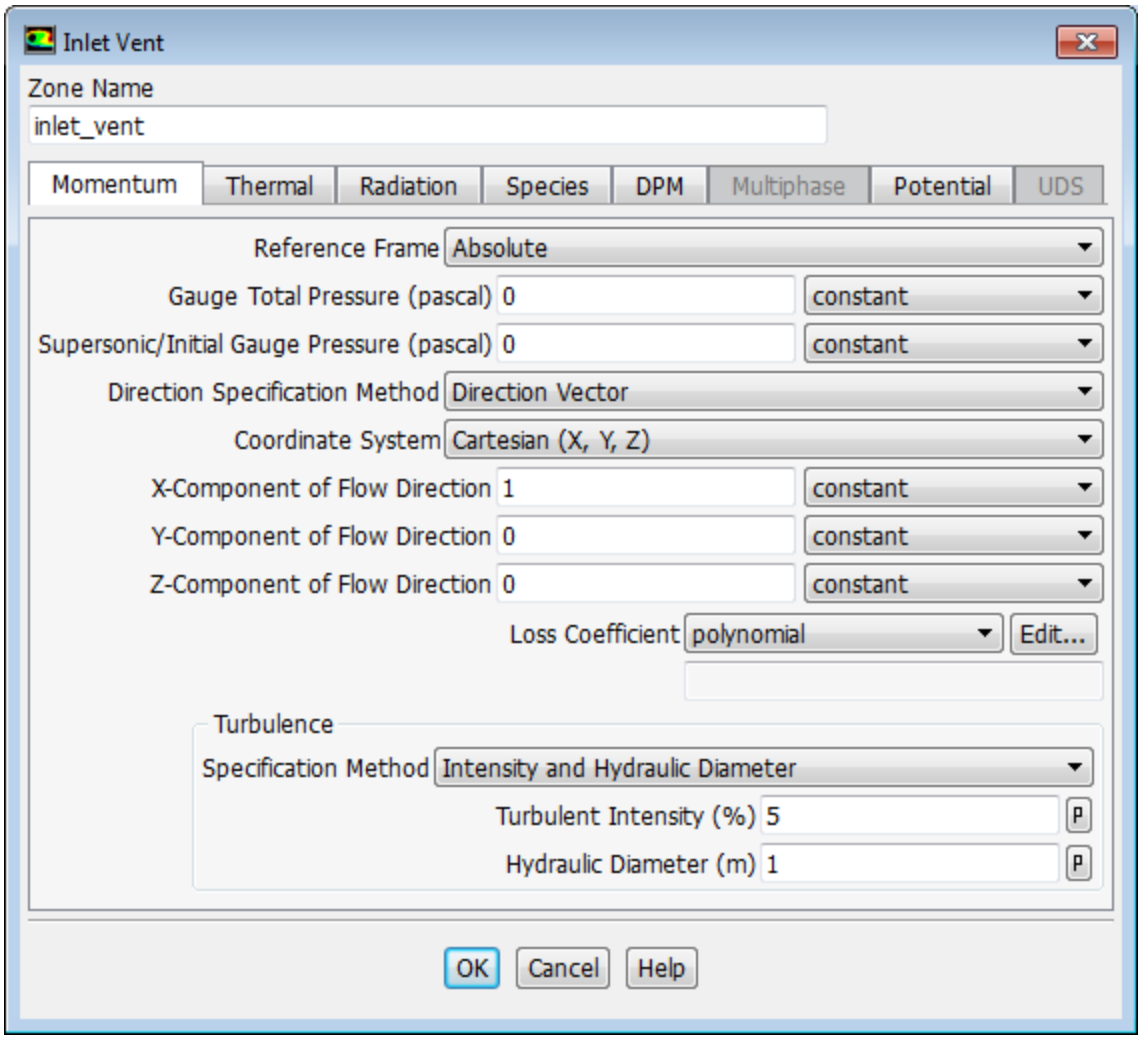
Radial Velocity Profile
contains a drop-down list from which you can select a boundary profile or a user-defined function for the definition of the radial velocity. This item will appear if you enable **Profile Specification of Radial Velocity**.

Radial-Velocity Polynomial Coefficients
sets the coefficients for the radial velocity polynomial. Separate the coefficients by spaces.

37.8.5. Inlet Vent Dialog Box

The **Inlet Vent** dialog box sets the boundary conditions for an inlet vent zone. It is opened from the Boundary Conditions Task Page (p. 2389). See Inputs at Inlet Vent Boundaries (p. 334) for details about defining the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone Name

sets the name of the zone.

Momentum

contains the momentum parameters.

Reference Frame

specifies the reference frame for the inlet vent. If the cell zone adjacent to the inlet vent is moving, you can choose to specify relative or absolute velocities by selecting **Relative to Adjacent Cell Zone** or **Absolute** in the **Reference Frame** drop-down list.

Gauge Total Pressure

sets the gauge total (or stagnation) pressure of the inflow stream. If you are using moving reference frames, see Defining Total Pressure and Temperature (p. 310) for information about relative and absolute total pressure.

Supersonic/Initial Gauge Pressure

sets the static pressure on the boundary when the flow becomes (locally) supersonic. It is also used to compute initial values for pressure, temperature, and velocity if the inlet vent boundary condition is selected for computing initial values (see Initializing the Entire Flow Field Using Standard Initialization (p. 1591)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Direction Specification Method
specifies the method you will use to define the flow direction. If you choose **Direction Vector**, you will define the flow direction components, and if you choose **Normal to Boundary** no inputs are required. See Defining the Flow Direction (p. 311) for information on specifying flow direction.

Coordinate System
specifies whether **Cartesian**, **Cylindrical**, **Local Cylindrical**, **Local Cylindrical Swirl** vector components will be specified. This item will appear only for 3D cases in which you have selected **Direction Vector** as the **Direction Specification Method**.

X,Y,Z-Component of Flow Direction
set the direction of the flow at the inlet boundary. These items will appear if the selected **Coordinate System** is **Cartesian** or the model is 2D non-axisymmetric.

Radial, Tangential, Axial Component of Flow Direction
set the direction of the flow at the inlet boundary. These items will appear for 2D axisymmetric cases, or for 3D cases for which the selected **Coordinate System** is **Cylindrical** or **Local Cylindrical**.

Axis Origin
sets the **X**, **Y**, and **Z** coordinates of the origin of the local cylindrical coordinate system.

Axis Direction
sets the **X**, **Y**, and **Z** components of the direction of the local cylindrical coordinate system.

Loss-Coefficient
sets the non-dimensional loss coefficient used to compute the pressure drop. See Specifying the Loss Coefficient (p. 335) for details.

Turbulence
lists the turbulence parameters.

Specification Method
specifies which method will be used to define the turbulence parameters. You can choose **K and Epsilon** (k- ε models and RSM only), **K and Omega** (k- ω models only), **Intensity and Length Scale**, **Intensity and Viscosity Ratio**, **Intensity and Hydraulic Diameter**, **Modified Turbulent Viscosity** (Spalart-Allmaras model only), or **Turbulent Viscosity Ratio** (Spalart-Allmaras model only). See Determining Turbulence Parameters (p. 303) for information about the inputs for each of these methods. (This item will appear only for turbulent flow calculations.)

Turbulent Kinetic Energy, Turbulent Dissipation Rate
set values for the turbulence kinetic energy k and its dissipation rate ε. These items will appear if you choose **K and Epsilon** as the **Specification Method**.

Turbulent Kinetic Energy, Specific Dissipation Rate
set values for the turbulence kinetic energy k and its specific dissipation rate ω. These items will appear if you choose **K and Omega** as the **Specification Method**.

Turbulence Intensity, Turbulence Length Scale
set values for turbulence intensity I and turbulence length scale ℓ. These items will appear if you choose **Intensity and Length Scale** as the **Specification Method**.

Turbulence Intensity, Turbulent Viscosity Ratio
set values for turbulence intensity I and turbulent viscosity ratio μ_t / μ. These items will appear if you choose **Intensity and Viscosity Ratio** as the **Specification Method**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Turbulence Intensity, Hydraulic Diameter
set values for turbulence intensity I and hydraulic diameter L. These items will appear if you choose **Intensity and Hydraulic Diameter** as the **Specification Method**.

Modified Turbulent Viscosity
sets the value of the modified turbulent viscosity $\tilde{\nu}$. This item will appear if you choose **Modified Turbulent Viscosity** as the **Specification Method**.

Turbulent Viscosity Ratio
sets the value of the turbulent viscosity ratio μ_t / μ. This item will appear if you choose **Turbulent Viscosity Ratio** as the **Specification Method**.

Reynolds-Stress Specification Method
specifies which method will be used to determine the Reynolds stress boundary conditions when the Reynolds stress turbulence model is used. You can choose either **K or Turbulence Intensity** or **Reynolds-Stress Components**. If you choose the former, ANSYS Fluent will compute the Reynolds stresses for you. If you choose the latter, you will explicitly specify the Reynolds stresses yourself. See Reynolds Stress Model (p. 808) for details. (This item will appear only for RSM turbulent flow calculations.)

UU, VV, WW, UV, VW, UW Reynolds Stresses
specify the Reynolds stress components when **Reynolds-Stress Components** is chosen as the **Reynolds-Stress Specification Method**.

Thermal
contains the thermal parameters.

Total Temperature
sets the total temperature of the inflow stream. If you are using moving reference frames, see Defining Total Pressure and Temperature (p. 310) for information about relative and absolute total temperature.

Radiation
contains the radiation parameters.

External Black Body Temperature Method, Internal Emissivity
set the radiation boundary conditions when you are using the P-1, DTRM, DO, S2S, or MC models for radiation heat transfer. See Defining Boundary Conditions for Radiation (p. 869) for details.

Participates in Solar Ray Tracing
specifies whether or not the inlet vent participates in solar ray tracing.

Solar Transmissivity Factor
specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the inlet vent.

Participates in View Factor Calculation
specifies whether or not the inlet vent participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

Species
contains the species parameters.

Specify Species in Mole Fractions
allows you to specify the species in mole fractions rather than mass fractions.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Species Mass Fractions
contains inputs for the mass fractions of defined species. See Defining Cell Zone and Boundary Conditions for Species (p. 990) for details about these inputs. (These items will appear only if you are modeling non-reacting multi-species flow or you are using the finite-rate reaction formulation.)

Mean Mixture Fraction, Mixture Fraction Variance
set inlet values for the PDF mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model.)

Secondary Mean Mixture Fraction, Secondary Mixture Fraction Variance
set inlet values for the secondary mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model with two mixture fractions.)

Progress Variable
sets the value of the progress variable for premixed turbulent combustion. See Setting Boundary Conditions for the Progress Variable (p. 1094) for details.

This item will appear only if the premixed or partially premixed combustion model is used.

DPM
contains the discrete phase parameters.

Discrete Phase BC Type
sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect
rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See Particle Reflection at Wall in the *Fluent Theory Guide*.)

trap
terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 23.25: "Trap" Boundary Condition for the Discrete Phase (p. 1303).

escape
reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 23.26: "Escape" Boundary Condition for the Discrete Phase (p. 1303).

wall-jet
indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 16.6: "Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

user-defined
specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function
sets the user-defined function from the drop-down list.

Multiphase
contains the multiphase parameters.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Granular Temperature
specifies temperature for the solids phase and is proportional to the kinetic energy of the random motion of the particles.

Volume Fraction
specifies the volume fraction of the secondary phase selected in the Boundary Conditions Task Page (p. 2389). This section of the dialog box will appear when one of the multiphase models is being used. See Defining Multiphase Cell Zone and Boundary Conditions (p. 1391) for details.

Open Channel
is available when the VOF model with open channel flow is enabled.

Secondary Phase for Inlet
is where the specified parameters are valid only for one secondary phase. In case of a three-phase flow, select the corresponding secondary phase from this list. This appears when **Open Channel** is enabled.

Flow Specification Method
allows you to select the type of flow. You can choose **Free Surface Level and Velocity**, **Total Height and Velocity**, or **Free Surface Level and Total Height**. This appears when **Open Channel** is enabled.

Free Surface Level
can be determined using the absolute value of height from the free surface to the origin in the direction of gravity, or by applying the correct sign based on whether the free surface level is above or below the origin.

Total Height
is used as an option for describing the flow. It is given by Equation 24.4 (p. 1414).

Bottom Level
is valid only for shallow waves. The bottom level is used for calculating the liquid height.

Velocity Magnitude
sets the magnitude of the velocity vector at the inflow boundary.

Level-Set Function Flux
appears if the **Level Set** option is enabled for the VOF model.

Potential
displays the boundary conditions for the electric potential field. This tab is available only if you have enabled either the **Electric Potential model** or the **Electrochemical** reaction model in the Species Dialog Box (p. 2332).

Potential Boundary Condition
is a drop-down list of available potential boundary condition types: **Specified Flux** and **Specified Value**. For the **Specified Flux** boundary condition, you will need to specify **Current Density** at the wall. For the **Specified Value** boundary condition, you will need to specify **Potential** at the wall.

UDS
contains the UDS parameters.

User-Defined Scalar Boundary Condition
appears only if user-defined scalars are specified.

User Scalar n
specifies whether the scalar is a specified flux or a specified value.

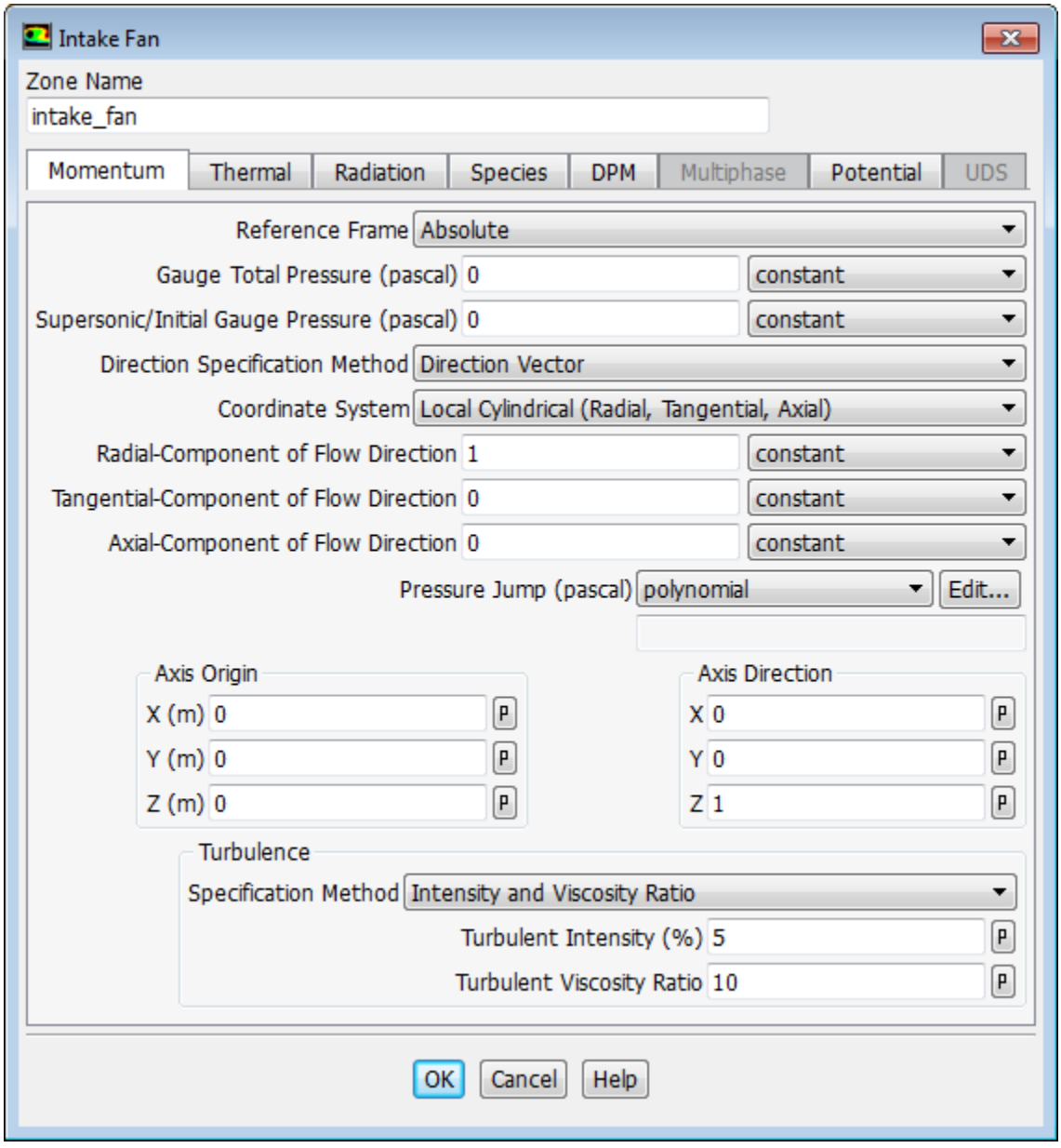
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

User-Defined Scalar Boundary Value
appears only if user-defined scalars are specified.

User Scalar n
specifies the value of the scalar.

37.8.6. Intake Fan Dialog Box

The **Intake Fan** dialog box sets the boundary conditions for an intake fan zone. It is opened from the Boundary Conditions Task Page (p. 2389). See Inputs at Intake Fan Boundaries (p. 336) for details about defining the items below.



Controls

Zone Name
sets the name of the zone.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Momentum
contains the momentum parameters.

Reference Frame
specifies the reference frame for the intake fan. If the cell zone adjacent to the intake fan is moving, you can choose to specify relative or absolute velocities by selecting **Relative to Adjacent Cell Zone** or **Absolute** in the **Reference Frame** drop-down list.

Gauge Total Pressure
sets the gauge total (or stagnation) pressure of the inflow stream. If you are using moving reference frames, see Defining Total Pressure and Temperature (p. 310) for information about relative and absolute total pressure.

Supersonic/Initial Gauge Pressure
sets the static pressure on the boundary when the flow becomes (locally) supersonic. It is also used to compute initial values for pressure, temperature, and velocity if the intake fan boundary condition is selected for computing initial values (see Initializing the Entire Flow Field Using Standard Initialization (p. 1591)).

Direction Specification Method
specifies the method you will use to define the flow direction. If you choose **Direction Vector**, you will define the flow direction components, and if you choose **Normal to Boundary** no inputs are required. See Defining the Flow Direction (p. 311) for information on specifying flow direction.

Coordinate System
specifies whether **Cartesian**, **Cylindrical**, **Local Cylindrical**,or **Local Cylindrical Swirl** vector components will be defined. This item will appear only for 3D cases in which you have selected **Direction Vector** as the **Direction Specification Method**.

X-, Y-, Z-Component of Flow Direction
set the direction of the flow at the inlet boundary. For compressible flow, if the inflow becomes supersonic, the velocity is not reoriented. These items will appear if the selected **Coordinate System** is **Cartesian** or the model is 2D non-axisymmetric.

Radial-, Tangential-, Axial-Component of Flow Direction
set the direction of the flow at the inlet boundary. For compressible flow, if the inflow becomes supersonic, the velocity is not reoriented. These items will appear for 2D axisymmetric cases, or for 3D cases for which the selected **Coordinate System** is **Cylindrical** or **Local Cylindrical**.

Pressure Jump
specifies the rise in pressure across the fan. See Specifying the Pressure Jump (p. 338) for details.

Axis Origin
sets the **X**, **Y**, and **Z** coordinates of the origin of the local cylindrical coordinate system.

Axis Direction
sets the **X**, **Y**, and **Z** components of the direction of the local cylindrical coordinate system.

Turbulence
consists of the turbulence parameters.

Specification Method
specifies which method will be used to define the turbulence parameters. You can choose **K and Epsilon** (k- ε models and RSM only), **K and Omega** (k- ω models only), **Intensity and Length**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scale, **Intensity and Viscosity Ratio**, **Intensity and Hydraulic Diameter**, **Modified Turbulent Viscosity** (Spalart-Allmaras model only), or **Turbulent Viscosity Ratio** (Spalart-Allmaras model only). See Determining Turbulence Parameters (p. 303) for information about the inputs for each of these methods. (This item will appear only for turbulent flow calculations.)

Turbulent Kinetic Energy, Turbulent Dissipation Rate
set values for the turbulence kinetic energy k and its dissipation rate ε. These items will appear if you choose **K and Epsilon** as the **Specification Method**.

Turbulent Kinetic Energy, Specific Dissipation Rate
set values for the turbulence kinetic energy k and its specific dissipation rate ω. These items will appear if you choose **K and Omega** as the **Specification Method**.

Turbulent Intensity, Turbulent Length Scale
set values for turbulence intensity I and turbulence length scale ℓ. These items will appear if you choose **Intensity and Length Scale** as the **Specification Method**.

Turbulent Intensity, Turbulent Viscosity Ratio
set values for turbulence intensity I and turbulent viscosity ratio μ_t / μ. These items will appear if you choose **Intensity and Viscosity Ratio** as the **Specification Method**.

Turbulent Intensity, Hydraulic Diameter
set values for turbulence intensity I and hydraulic diameter L. These items will appear if you choose **Intensity and Hydraulic Diameter** as the **Specification Method**.

Modified Turbulent Viscosity
sets the value of the modified turbulent viscosity $\tilde{\nu}$. This item will appear if you choose **Modified Turbulent Viscosity** as the **Specification Method**.

Turbulent Viscosity Ratio
sets the value of the turbulent viscosity ratio μ_t / μ. This item will appear if you choose **Turbulent Viscosity Ratio** as the **Specification Method**.

Reynolds-Stress Specification Method
specifies which method will be used to determine the Reynolds stress boundary conditions when the Reynolds stress turbulence model is used. You can choose either **K or Turbulent Intensity** or **Reynolds-Stress Components**. If you choose the former, ANSYS Fluent will compute the Reynolds stresses for you. If you choose the latter, you will explicitly specify the Reynolds stresses yourself. See Reynolds Stress Model (p. 808) for details. (This item will appear only for RSM turbulent flow calculations.)

UU, VV, WW, UV, VW, UW Reynolds Stresses
specify the Reynolds stress components when **Reynolds-Stress Components** is chosen as the **Reynolds-Stress Specification Method**.

Thermal
contains the thermal parameters.

Total Temperature
sets the total temperature of the inflow stream. If you are using moving reference frames, see Defining Total Pressure and Temperature (p. 310) for information about relative and absolute total temperature.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Radiation
contains the radiation parameters.

External Black Body Temperature Method, Internal Emissivity
set the radiation boundary conditions when you are using the P-1, DTRM, DO, S2S, or MC models for radiation heat transfer. See Defining Boundary Conditions for Radiation (p. 869) for details.

Participates in Solar Ray Tracing
specifies whether or not the intake fan participates in solar ray tracing.

Solar Transmissivity Factor
specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the intake fan.

Participates in View Factor Calculation
specifies whether or not the intake fan participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

Species
contains the species parameters.

Specify Species in Mole Fractions
allows you to specify the species in mole fractions rather than mass fractions.

Species Mass Fractions
contains inputs for the mass fractions of defined species. See Defining Cell Zone and Boundary Conditions for Species (p. 990) for details about these inputs. (These items will appear only if you are modeling non-reacting multi-species flow or you are using the finite-rate reaction formulation.)

Mean Mixture Fraction, Mixture Fraction Variance
set inlet values for the PDF mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model.)

Secondary Mean Mixture Fraction, Secondary Mixture Fraction Variance
set inlet values for the secondary mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model with two mixture fractions.)

Progress Variable
sets the value of the progress variable for premixed turbulent combustion. See Setting Boundary Conditions for the Progress Variable (p. 1094) for details.

This item will appear only if the premixed or partially premixed combustion model is used.

DPM
contains the discrete phase parameters.

Discrete Phase BC Type
sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect
rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See Particle Reflection at Wall in the *Fluent Theory Guide*.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

trap
terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 23.25: "Trap" Boundary Condition for the Discrete Phase (p. 1303).

escape
reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 23.26: "Escape" Boundary Condition for the Discrete Phase (p. 1303).

wall-jet
indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 16.6: "Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

user-defined
specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function
sets the user-defined function from the drop-down list.

Multiphase
contains the multiphase parameters.

Granular Temperature
specifies temperature for the solids phase and is proportional to the kinetic energy of the random motion of the particles.

Volume Fraction
specifies the volume fraction of the secondary phase selected in the Boundary Conditions Task Page (p. 2389). This section of the dialog box will appear when one of the multiphase models is being used. See Defining Multiphase Cell Zone and Boundary Conditions (p. 1391) for details.

Potential
displays the boundary conditions for the electric potential field. This tab is available only if you have enabled either the **Electric Potential model** or the **Electrochemical** reaction model in the Species Dialog Box (p. 2332).

Potential Boundary Condition
is a drop-down list of available potential boundary condition types: **Specified Flux** and **Specified Value**. For the **Specified Flux** boundary condition, you will need to specify **Current Density** at the wall. For the **Specified Value** boundary condition, you will need to specify **Potential** at the wall.

UDS
contains the UDS parameters.

User-Defined Scalar Boundary Condition
appears only if user-defined scalars are specified.

User Scalar n
specifies whether the scalar is a specified flux or a specified value.

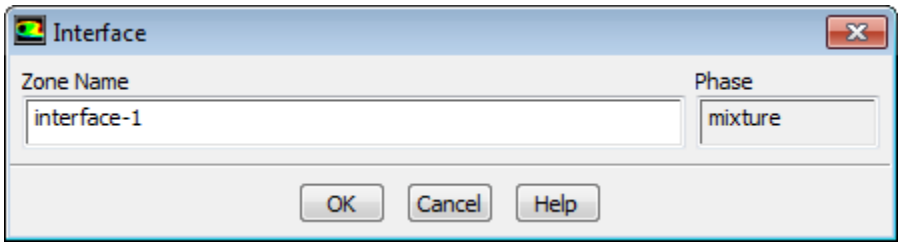
User-Defined Scalar Boundary Value
appears only if user-defined scalars are specified.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

User Scalar n
specifies the value of the scalar.

37.8.7. Interface Dialog Box

The **Interface** dialog box can be used to modify the name of an interface zone; there are no conditions to be set. It is opened from the Boundary Conditions Task Page (p. 2389). Interface zones are used for multiple reference frame and sliding mesh calculations, and for non-conformal meshes. See The Multiple Reference Frame Model (p. 613), Setting Up the Sliding Mesh Problem (p. 634), and Non-Conformal Meshes (p. 150) for details.



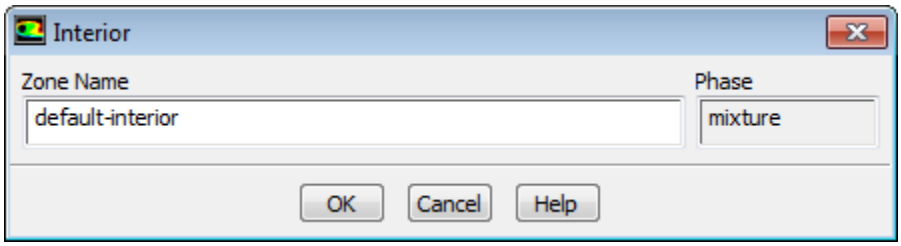
Controls

Zone Name
sets the name of the zone.

Phase
displays the name of the phase. This item appears only for multiphase flows.

37.8.8. Interior Dialog Box

The **Interior** dialog box can be used to modify the name of an interior zone; there are no conditions to be set. It is opened from the Boundary Conditions Task Page (p. 2389).



Controls

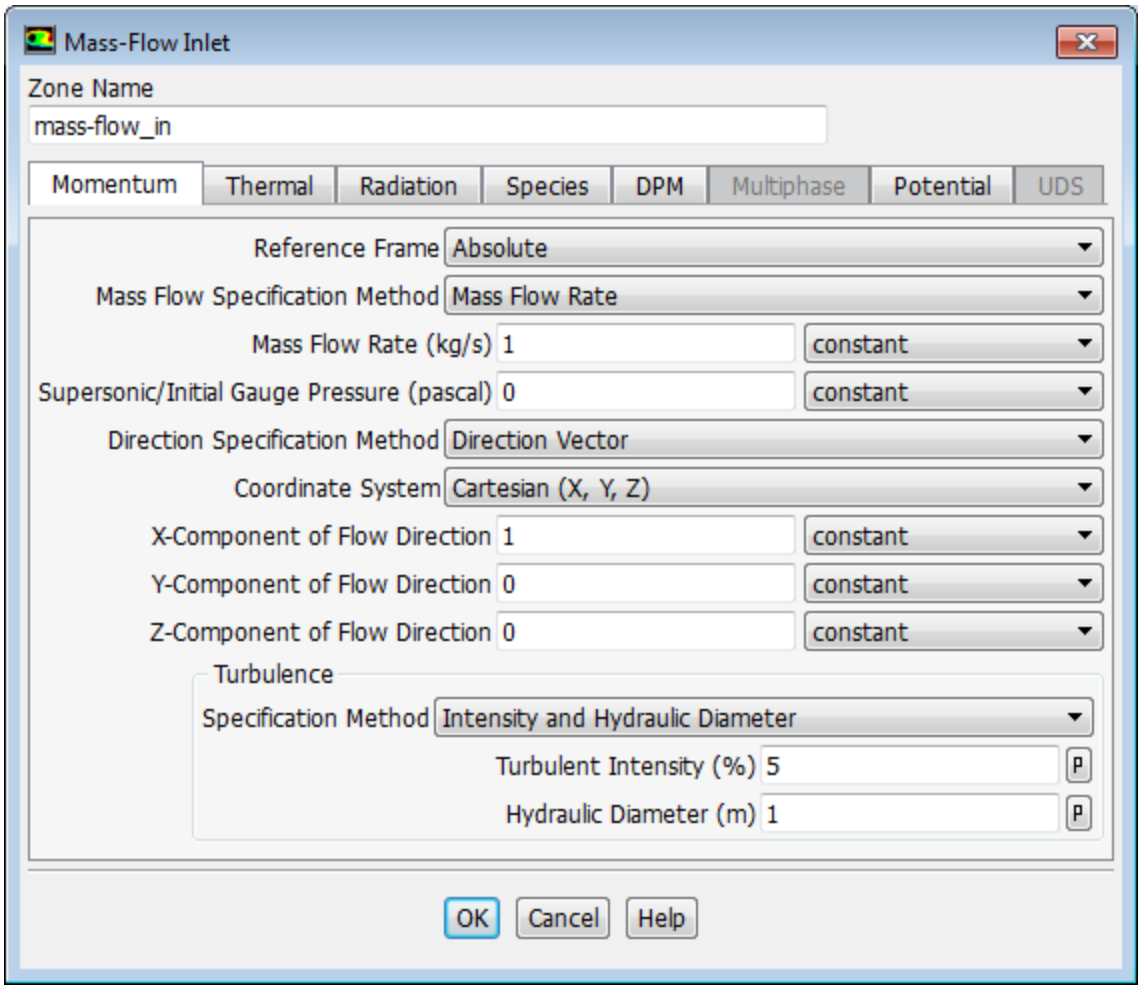
Zone Name
sets the name of the zone.

Phase
displays the name of the phase. This item appears only for multiphase flows.

37.8.9. Mass-Flow Inlet Dialog Box

The **Mass-Flow Inlet** dialog box sets the boundary conditions for a mass-flow inlet zone. It is opened from the Boundary Conditions Task Page (p. 2389). See Inputs at Mass-Flow Inlet Boundaries (p. 323) for details about defining the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone Name
sets the name of the zone.

Momentum
displays the momentum boundary conditions.

Reference Frame
specifies the reference frame for the mass flow. If the cell zone adjacent to the mass-flow inlet is moving, you can choose to specify relative or absolute velocities by selecting **Relative to Adjacent Cell Zone** or **Absolute** in the **Reference Frame** drop-down list.

Mass Flow Specification Method
specifies whether you are defining **Mass Flow Rate**, **Mass Flux**, or **Mass Flux with Average Mass Flux**.

Mass Flow Rate
sets the prescribed mass flow rate for the zone. This flow rate is converted internally to a prescribed uniform mass flux over the zone by dividing the flow rate by the flow direction area projection of the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

zone. This item will appear if you selected **Mass Flow Rate** in the **Mass Flow Specification Method** list.

Important

Note that for axisymmetric problems, this mass flow rate is the flow rate through the entire (2π-radian) domain, not through a 1-radian slice.

Mass Flux

sets the prescribed mass flux for the zone. This item will appear if you selected **Mass Flux** or **Mass Flux with Average Mass Flux** in the **Mass Flow Specification Method** list.

Important

Note that for axisymmetric problems, this mass flux is the flux through a 1-radian slice of the domain.

Average Mass Flux

sets the average mass flux through the zone. See More About Mass Flux and Average Mass Flux (p. 325) for details. This item will appear if you selected **Mass Flux with Average Mass Flux** in the **Mass Flow Specification Method** list.

Important

Note that for axisymmetric problems, this mass flux is the flux through a 1-radian slice of the domain.

Supersonic/Initialization Gauge Pressure

sets the static pressure that will be used to initialize the flow field if the mass flow inlet boundary condition is selected for initializing flow properties (see Initializing the Entire Flow Field Using Standard Initialization (p. 1591)).

Direction Specification Method

specifies the method you will use to define the flow direction. If you choose **Direction Vector**, you will define the flow direction components, and if you choose **Normal to Boundary** no inputs are required. See Defining the Flow Direction (p. 311) for information on specifying flow direction.

Coordinate System

specifies whether **Cartesian**, **Cylindrical**, **Local Cylindrical**,or **Local Cylindrical Swirl** vector components will be defined. This item will appear only for 3D cases in which you have selected **Direction Vector** as the **Direction Specification Method**.

X-, Y-, Z-Component of Flow Direction

set the velocity-direction vector of the inflow stream. This vector does not need to be normalized (for example, you can specify the vector (1 1 1) rather than (0.577 0.577 0.577)). These items will appear if the selected **Coordinate System** is **Cartesian** or the model is 2D non-axisymmetric.

Radial-, Tangential-, Axial-Component of Flow Direction

set the velocity-direction vector of the inflow stream. These items will appear for 2D axisymmetric cases, or for 3D cases for which the selected **Coordinate System** is **Cylindrical** or **Local Cylindrical**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Axial-, Radial-Component of Flow Direction, Tangential-Velocity
appear for a 3D **Local Cylindrical Swirl** coordinate system.

Axis Origin
sets the **X**, **Y**, and **Z** coordinates of the origin of the local cylindrical (swirl) coordinate system.

Axis Direction
sets the **X**, **Y**, and **Z** components of the direction of the local cylindrical (swirl) coordinate system.

Turbulence
contains the turbulence parameters.

Specification Method
specifies which method will be used to define the turbulence parameters. You can choose **K and Epsilon** (k- ε models and RSM only), **K and Omega** (k- ω models only), **Intensity and Length Scale**, **Intensity and Viscosity Ratio**, **Intensity and Hydraulic Diameter**, **Modified Turbulent Viscosity** (Spalart-Allmaras model only), or **Turbulent Viscosity Ratio** (Spalart-Allmaras model only). See Determining Turbulence Parameters (p. 303) for information about the inputs for each of these methods. (This item will appear only for turbulent flow calculations.)

Turbulent Kinetic Energy, Turbulent Dissipation Rate
set values for the turbulence kinetic energy k and its dissipation rate ε. These items will appear if you choose **K and Epsilon** as the **Specification Method**.

Turbulent Kinetic Energy, Specific Dissipation Rate
set values for the turbulence kinetic energy k and its specific dissipation rate ω. These items will appear if you choose **K and Omega** as the **Specification Method**.

Turbulent Intensity, Turbulent Length Scale
set values for turbulence intensity I and turbulence length scale ℓ. These items will appear if you choose **Intensity and Length Scale** as the **Specification Method**.

Turbulent Intensity, Turbulent Viscosity Ratio
set values for turbulence intensity I and turbulent viscosity ratio μ_t / μ. These items will appear if you choose **Intensity and Viscosity Ratio** as the **Specification Method**.

Turbulent Intensity, Hydraulic Diameter
set values for turbulence intensity I and hydraulic diameter L. These items will appear if you choose **Intensity and Hydraulic Diameter** as the **Specification Method**.

Modified Turbulent Viscosity
sets the value of the modified turbulent viscosity $\tilde{\nu}$. This item will appear if you choose **Modified Turbulent Viscosity** as the **Specification Method**.

Turbulent Viscosity Ratio
sets the value of the turbulent viscosity ratio μ_t / μ. This item will appear if you choose **Turbulent Viscosity Ratio** as the **Specification Method**.

Reynolds-Stress Specification Method
specifies which method will be used to determine the Reynolds stress boundary conditions when the Reynolds stress turbulence model is used. You can choose either **K or Turbulent Intensity** or **Reynolds-Stress Components**. If you choose the former, ANSYS Fluent will compute the Reynolds stresses for you. If you choose the latter, you will explicitly specify the Reynolds stresses yourself.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See Reynolds Stress Model (p. 808) for details. (This item will appear only for RSM turbulent flow calculations.)

UU, VV, WW, UV, VW, UW Reynolds Stresses
specify the Reynolds stress components when **Reynolds-Stress Components** is chosen as the **Reynolds-Stress Specification Method**.

Acoustic Wave Model
contains settings for treatment of acoustic pressure waves at the boundary.

Off
disables special treatment of acoustic pressure waves at the boundary.

Non Reflecting
enables the general non-reflecting boundary condition treatment described in General Non-Reflecting Boundary Conditions (p. 416).

Impedance
enables the impedance boundary condition treatment described in Impedance Boundary Conditions (p. 422).

Transparent Flow Forcing
enables the transparent flow forcing boundary condition described in Transparent Flow Forcing Boundary Conditions (p. 425).

Impedance Parameters
contains the parameters for the impedance boundary condition treatment. For details refer to Using the Impedance Boundary Condition (p. 424).

Transparent Flow Forcing Parameters
contains the parameters for the transparent flow forcing boundary condition treatment. For details refer to Using the Transparent Flow Forcing Boundary Condition (p. 426).

Thermal
contains the thermal parameters.

Total Temperature
sets the total temperature of the inflow stream.

Radiation
contains the radiation parameters.

External Black Body Temperature Method, Internal Emissivity
set the radiation boundary conditions when you are using the P-1, DTRM, DO, S2S, or MC models for radiation heat transfer. See Defining Boundary Conditions for Radiation (p. 869) for details.

Participates in Solar Ray Tracing
specifies whether or not the mass-flow inlet participates in solar ray tracing.

Solar Transmissivity Factor
specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the mass-flow inlet.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Participates in View Factor Calculation
specifies whether or not the mass-flow inlet participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

Species
contains the species parameters.

Specify Species in Mole Fractions
allows you to specify the species in mole fractions rather than mass fractions.

Species Mass Fractions
contains inputs for the mass fractions of defined species. See Defining Cell Zone and Boundary Conditions for Species (p. 990) for details about these inputs. These items will appear only if you are modeling non-reacting multi-species flow or you are using the finite-rate reaction formulation.

Progress Variable
sets the value of the progress variable for premixed turbulent combustion. See Setting Boundary Conditions for the Progress Variable (p. 1094) for details.

This item will appear only if the premixed or partially premixed combustion model is used.

Mean Mixture Fraction, Mixture Fraction Variance
set inlet values for the PDF mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model.)

Secondary Mean Mixture Fraction, Secondary Mixture Fraction Variance
set inlet values for the secondary mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model with two mixture fractions.)

DPM
contains the discrete phase parameters.

Discrete Phase BC Type
sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect
rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See Particle Reflection at Wall in the *Fluent Theory Guide*.)

trap
terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 23.25: "Trap" Boundary Condition for the Discrete Phase (p. 1303).

escape
reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 23.26: "Escape" Boundary Condition for the Discrete Phase (p. 1303).

wall-jet
indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 16.6: "Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

user-defined
specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function
sets the user-defined function from the drop-down list.

Potential
displays the boundary conditions for the electric potential field. This tab is available only if you have enabled either the **Electric Potential model** or the **Electrochemical** reaction model in the Species Dialog Box (p. 2332).

Potential Boundary Condition
is a drop-down list of available potential boundary condition types: **Specified Flux** and **Specified Value**. For the **Specified Flux** boundary condition, you will need to specify **Current Density** at the wall. For the **Specified Value** boundary condition, you will need to specify **Potential** at the wall.

UDS
contains the UDS parameters.

User-Defined Scalar Boundary Condition
appears only if user-defined scalars are specified.

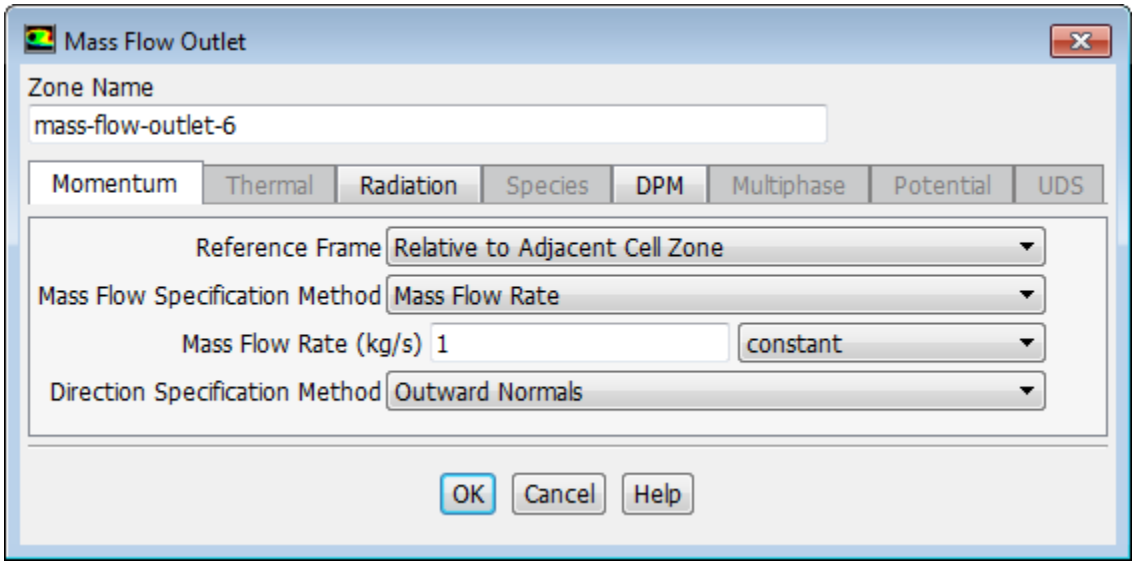
User Scalar n
specifies whether the scalar is a specified flux or a specified value.

User-Defined Scalar Boundary Value
appears only if user-defined scalars are specified.

User Scalar n
specifies the value of the scalar.

37.8.10. Mass Flow Outlet Dialog Box

The **Mass Flow Outlet** dialog box sets the boundary conditions for a mass-flow outlet zone. It is opened from the Boundary Conditions Task Page (p. 2389). See Inputs at Mass-Flow Outlet Boundaries (p. 330) for details about defining the items below.



Controls

Zone Name
sets the name of the zone.

Momentum
displays the momentum boundary conditions.

Reference Frame
specifies the reference frame for the mass flow when the cell zone adjacent to the mass-flow outlet is moving. The only available option is **Relative to Adjacent Cell Zone**, so you must define the mass flow relative to the adjacent cell zone.

Mass Flow Specification Method
specifies whether you are defining **Mass Flow Rate**, **Mass Flux**, or **Mass Flux with Average Mass Flux**.

Mass Flow Rate
sets the prescribed mass flow rate for the zone. This flow rate is converted internally to a prescribed uniform mass flux over the zone by dividing the flow rate by the flow direction area projection of the zone. This item will appear if you selected **Mass Flow Rate** from the **Mass Flow Specification Method** drop-down list.

Important

Note that for axisymmetric problems, this mass flow rate is the flow rate through the entire (2π-radian) domain, not through a 1-radian slice.

Mass Flux
sets the prescribed mass flux for the zone. This item will appear if you selected **Mass Flux** or **Mass Flux with Average Mass Flux** from the **Mass Flow Specification Method** drop-down list.

Important

Note that for axisymmetric problems, this mass flux is the flux through a 1-radian slice of the domain.

Average Mass Flux
sets the average mass flux through the zone. See More About Mass Flux and Average Mass Flux (p. 332) for details. This item will appear if you selected **Mass Flux with Average Mass Flux** from the **Mass Flow Specification Method** drop-down list.

Important

Note that for axisymmetric problems, this mass flux is the flux through a 1-radian slice of the domain.

Direction Specification Method
specifies the method you will use to define the flow direction. If you choose **Direction Vector**, you will define the flow direction components; if you choose **Outward Normals**, the flow direction will be out of the domain normal to boundary (and no further inputs are required). See Defining the Flow Direction (p. 332) for information on specifying flow direction.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coordinate System
specifies whether **Cartesian**, **Cylindrical**, **Local Cylindrical**,or **Local Cylindrical Swirl** vector components will be defined. This item will appear only for 3D cases in which you have selected **Direction Vector** as the **Direction Specification Method**.

X-, Y-, Z-Component of Flow Direction
set the velocity-direction vector of the outflow stream. This vector does not need to be normalized (for example, you can specify the vector (1 1 1) rather than (0.577 0.577 0.577)). These items will appear if the selected **Coordinate System** is **Cartesian** or the model is 2D non-axisymmetric.

Radial-, Tangential-, Axial-Component of Flow Direction
set the velocity-direction vector of the outflow stream. These items will appear for 2D axisymmetric cases, or for 3D cases for which the selected **Coordinate System** is **Cylindrical** or **Local Cylindrical**.

Axial-, Radial-Component of Flow Direction, Tangential-Velocity
appear for a 3D **Local Cylindrical Swirl** coordinate system.

Axis Origin
sets the **X**, **Y**, and **Z** coordinates of the origin of the local cylindrical (swirl) coordinate system.

Axis Direction
sets the **X**, **Y**, and **Z** components of the direction of the local cylindrical (swirl) coordinate system.

Radiation
contains the radiation parameters.

External Black Body Temperature Method, Black Body Temperature, Internal Emissivity
set the radiation boundary conditions when you are using the P-1, DTRM, DO, S2S, or MC models for radiation heat transfer. See Defining Boundary Conditions for Radiation (p. 869) for details.

Participates in Solar Ray Tracing
specifies whether or not the mass-flow outlet participates in solar ray tracing.

Solar Transmissivity Factor
specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the mass-flow outlet.

Participates in View Factor Calculation
specifies whether or not the mass-flow outlet participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

DPM
contains the discrete phase parameters.

Discrete Phase BC Type
sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect
rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See Particle Reflection at Wall in the *Fluent Theory Guide*.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

trap
terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 23.25: "Trap" Boundary Condition for the Discrete Phase (p. 1303).

escape
reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 23.26: "Escape" Boundary Condition for the Discrete Phase (p. 1303).

wall-jet
indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 16.6: "Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

user-defined
specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function
sets the user-defined function from the drop-down list.

UDS
contains the UDS parameters.

User-Defined Scalar Boundary Condition
appears only if user-defined scalars are specified.

User Scalar n
specifies whether the scalar is a specified flux or a specified value.

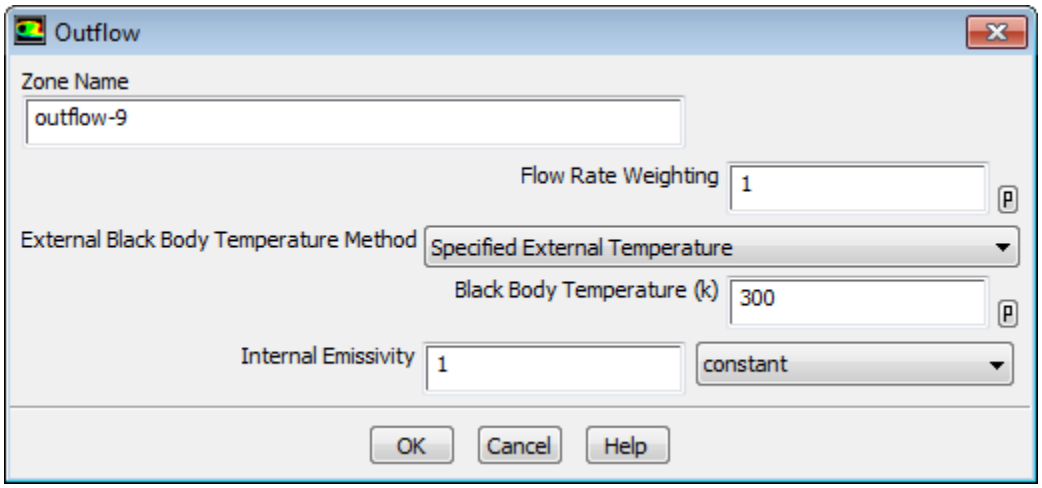
User-Defined Scalar Boundary Value
appears only if user-defined scalars are specified.

User Scalar n
specifies the value of the scalar.

37.8.11. Outflow Dialog Box

The **Outflow** dialog box sets the boundary conditions for an outflow zone. It is opened from the Boundary Conditions Task Page (p. 2389). See Using Outflow Boundaries (p. 351) for details about using outflow boundaries.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone Name
sets the name of the zone.

Flow Rate Weighting
specifies the portion of the outflow that is going through the boundary. See Mass Flow Split Boundary Conditions (p. 352) for details.

External Black Body Temperature Method, Internal Emissivity
set the radiation boundary conditions when you are using the P-1, DTRM, DO, S2S, or MC models for radiation heat transfer. See Defining Boundary Conditions for Radiation (p. 869) for details.

Discrete Phase BC Type
sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect
rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See Particle Reflection at Wall in the *Fluent Theory Guide*.)

trap
terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 23.25: "Trap" Boundary Condition for the Discrete Phase (p. 1303).

escape
reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 23.26: "Escape" Boundary Condition for the Discrete Phase (p. 1303).

wall-jet
indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 16.6: "Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

user-defined
specifies a user-defined function to define the discrete phase boundary condition type.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Participates in Solar Ray Tracing
specifies whether or not outflow participate in solar ray tracing.

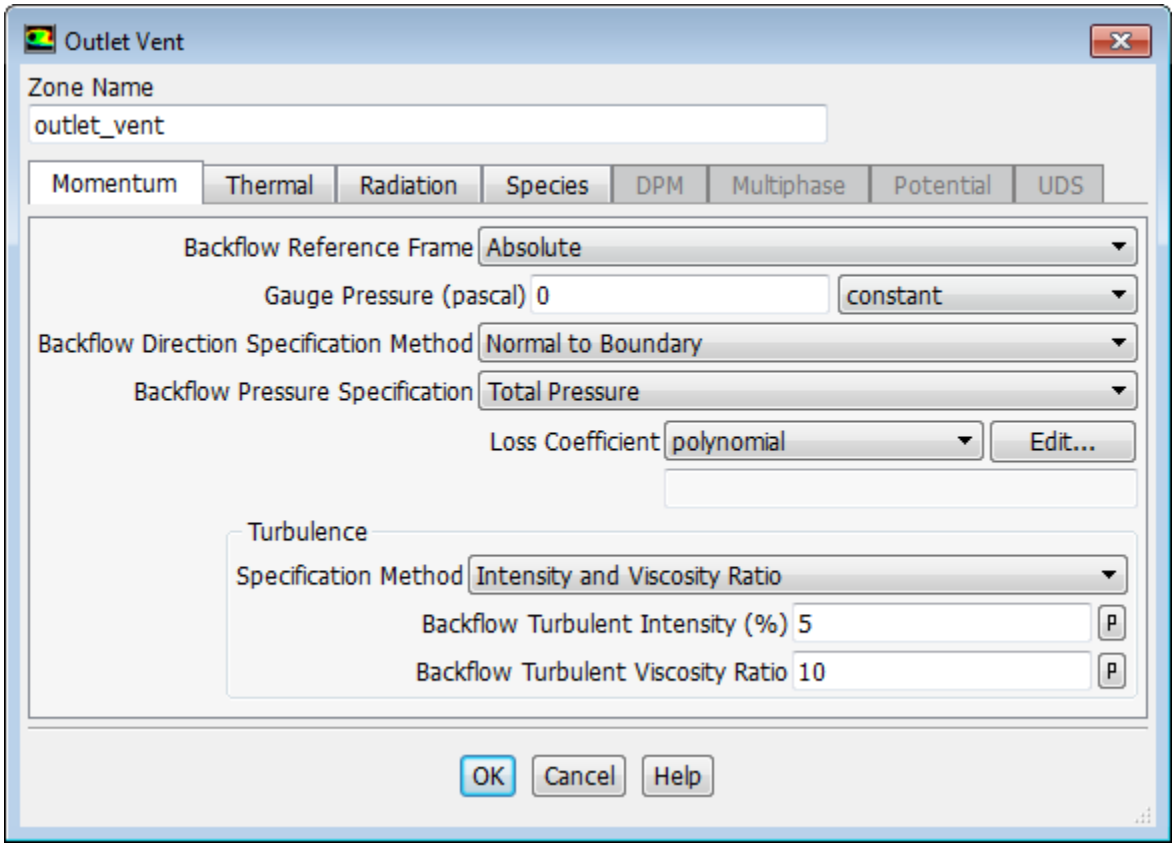
Solar Transmissivity Factor
specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the outflow.

Discrete Phase BC Function
sets the user-defined function from the drop-down list.

Participates in View Factor Calculation
specifies whether or not the outflow participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

37.8.12. Outlet Vent Dialog Box

The **Outlet Vent** dialog box sets the boundary conditions for an outlet vent zone. It is opened from the Boundary Conditions Task Page (p. 2389). See Inputs at Outlet Vent Boundaries (p. 353) for details about defining the items below.



Controls

Zone Name
sets the name of the zone.

Momentum
contains the momentum parameters.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Backflow Reference Frame
specify whether backflow temperature, pressure, and flow directions are in the **Absolute** or **Relative to the Adjacent Cell Zone** reference frame.

Gauge Pressure
sets the gauge pressure at the outlet boundary.

Backflow Direction Specification Method
specifies the method you will use to define the flow direction. If you choose **Direction Vector**, you will define the flow direction components, and if you choose **Normal to Boundary** or **From Neighboring Cell** no inputs are required. See Defining the Flow Direction (p. 311) for information on specifying flow direction.

Coordinate System
contains a drop-down list for selecting the coordinate system. You can choose **Cartesian**, **Cylindrical**, or **Local Cylindrical**. This option is available only when **Direction Vector** is selected from the **Backflow Direction Specification Method** drop-down list.

X-, Y-, Z-Component of Flow Direction
allows you to specify the velocity components in x, y, and z directions respectively. This option is available when **Cartesian** is selected for the **Coordinate System**.

Radial-, Tangential-, Axial-Component of Flow Direction
set the direction of the flow at the boundary. These items will appear for 2D axisymmetric cases, or for 3D cases for which the selected **Coordinate System** is **Cylindrical** or **Local Cylindrical**.

Backflow Pressure Specification
specifies how the pressure is calculated under backflow conditions. If you select **Static Pressure**, the **Gauge Pressure** is directly imposed as the boundary face pressure; if you select **Total Pressure**, the **Gauge Pressure** will be combined with a dynamic contribution that is based on the velocity in the adjacent cell zone.

Loss-Coefficient
sets the non-dimensional loss coefficient used to compute the pressure drop. See Specifying the Loss Coefficient (p. 355) for details.

Axis Origin
sets the **X**, **Y**, and **Z** coordinates of the origin of the local cylindrical coordinate system.

Axis Direction
sets the **X**, **Y**, and **Z** components of the direction of the local cylindrical coordinate system.

Turbulence
contains the turbulence parameters.

Specification Method
specifies which method will be used to define the turbulence parameters. You can choose **K and Epsilon** (k- ε models and RSM only), **K and Omega** (k- ω models only), **Intensity and Length Scale**, **Intensity and Viscosity Ratio**, **Intensity and Hydraulic Diameter**, **Modified Turbulent Viscosity** (Spalart-Allmaras model only), or **Turbulent Viscosity Ratio** (Spalart-Allmaras model only). See Determining Turbulence Parameters (p. 303) for information about the inputs for each of these methods. (This item will appear only for turbulent flow calculations.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Backflow Turbulent Kinetic Energy, Backflow Turbulent Dissipation Rate
set values for the turbulence kinetic energy k and its dissipation rate ε. These items will appear if you choose **K and Epsilon** as the **Specification Method**.

Backflow Turbulent Kinetic Energy, Backflow Specific Dissipation Rate
set values for the turbulence kinetic energy k and its specific dissipation rate ω. These items will appear if you choose **K and Omega** as the **Specification Method**.

Backflow Turbulent Intensity, Backflow Turbulent Length Scale
set values for turbulence intensity I and turbulence length scale ℓ. These items will appear if you choose **Intensity and Length Scale** as the **Specification Method**.

Backflow Turbulent Intensity, Backflow Turbulent Viscosity Ratio
set values for turbulence intensity I and turbulent viscosity ratio μ_t / μ. These items will appear if you choose **Intensity and Viscosity Ratio** as the **Specification Method**.

Backflow Turbulent Intensity, Backflow Hydraulic Diameter
set values for turbulence intensity I and hydraulic diameter L. These items will appear if you choose **Intensity and Hydraulic Diameter** as the **Specification Method**.

Backflow Modified Turbulent Viscosity
sets the value of the backflow modified turbulent viscosity $\tilde{\nu}$. This item will appear if you choose **Modified Turbulent Viscosity** as the **Specification Method**.

Backflow Turbulent Viscosity Ratio
sets the value of the backflow turbulent viscosity ratio μ_t / μ. This item will appear if you choose **Turbulent Viscosity Ratio** as the **Specification Method**.

Reynolds-Stress Specification Method
specifies which method will be used to determine the backflow Reynolds stress boundary conditions when the Reynolds stress turbulence model is used. You can choose either **K or Turbulent Intensity** or **Reynolds-Stress Components**. If you choose the former, ANSYS Fluent will compute the Reynolds stresses for you. If you choose the latter, you will explicitly specify the Reynolds stresses yourself. See Reynolds Stress Model (p. 808) for details. (This item will appear only for RSM turbulent flow calculations.)

Backflow UU, VV, WW, UV, VW, UW Reynolds Stresses
specify the backflow Reynolds stress components when **Reynolds-Stress Components** is chosen as the **Reynolds-Stress Specification Method**.

Thermal
contains the thermal parameters.

Backflow Total Temperature
sets the total temperature of the inflow stream should the flow reverse direction

Radiation
contains the boundary conditions for the radiation model at the outlet vent.

External Black Body Temperature Method, Internal Emissivity
set the radiation boundary conditions when you are using the P-1, DTRM, DO, S2S, or MC models for radiation heat transfer. See Defining Boundary Conditions for Radiation (p. 869) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Participates in Solar Ray Tracing
specifies whether or not the outlet vent participates in solar ray tracing.

Solar Transmissivity Factor
specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the outlet vent.

Participates in View Factor Calculation
specifies whether or not the outlet vent participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

Species
contains the species parameters.

Specify Species in Mole Fractions
allows you to specify the species in mole fractions rather than mass fractions.

Mean Mixture Fraction, Mixture Fraction Variance
set inlet values for the PDF mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model.)

Secondary Mean Mixture Fraction, Secondary Mixture Fraction Variance
set inlet values for the secondary mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model with two mixture fractions.)

Species Mass Fractions
contains inputs for the mass fractions of defined species. See Defining Cell Zone and Boundary Conditions for Species (p. 990) for details about these inputs. These items will appear only if you are modeling non-reacting multi-species flow or you are using the finite-rate reaction formulation.

Backflow Progress Variable
sets the value of the progress variable for premixed turbulent combustion. See Setting Boundary Conditions for the Progress Variable (p. 1094) for details.

This item will appear only if the premixed or partially premixed combustion model is used.

DPM
contains the discrete phase parameters.

Discrete Phase BC Type
sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect
rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See Particle Reflection at Wall in the *Fluent Theory Guide*.)

trap
terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 23.25: "Trap" Boundary Condition for the Discrete Phase (p. 1303).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

escape
reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 23.26: "Escape" Boundary Condition for the Discrete Phase (p. 1303).

wall-jet
indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 16.6: "Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

user-defined
specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function
sets the user-defined function from the drop-down list.

Multiphase
contains the multiphase parameters.

Backflow Granular Temperature
specifies temperature for the solids phase and is proportional to the kinetic energy of the random motion of the particles.

Volume Fraction Specification Method
sets the method used to specify the volume fraction of the secondary phase selected in the Boundary Conditions Task Page (p. 2389). This section of the dialog box will appear when one of the multiphase models is being used. See Defining Multiphase Cell Zone and Boundary Conditions (p. 1391) for details.

Backflow Volume Fraction
specifies the volume fraction of the secondary phase as a constant, profile, of UDF function.

From Neighboring Cell
calculates the volume fraction from the neighboring cells.

Potential
displays the boundary conditions for the electric potential field. This tab is available only if you have enabled either the **Electric Potential model** or the **Electrochemical** reaction model in the Species Dialog Box (p. 2332).

Potential Boundary Condition
is a drop-down list of available potential boundary condition types: **Specified Flux** and **Specified Value**. For the **Specified Flux** boundary condition, you will need to specify **Current Density** at the wall. For the **Specified Value** boundary condition, you will need to specify **Potential** at the wall.

UDS
contains the UDS parameters.

User-Defined Scalar Boundary Condition
appears only if user-defined scalars are specified.

User Scalar n
specifies whether the scalar is a specified flux or a specified value.

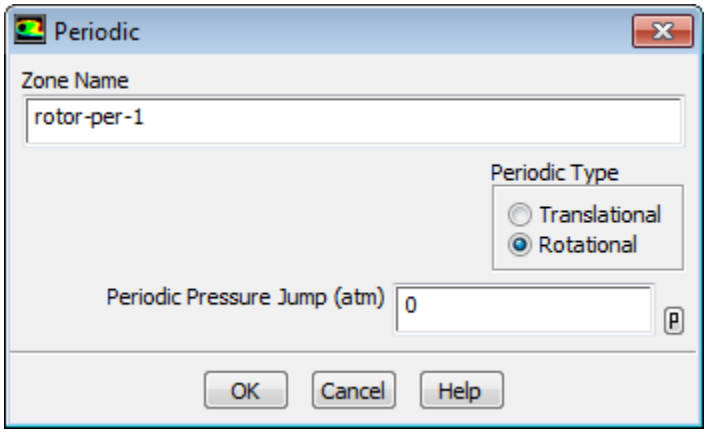
User-Defined Scalar Boundary Value
appears only if user-defined scalars are specified.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

User Scalar n
specifies the value of the scalar.

37.8.13. Periodic Dialog Box

The **Periodic** dialog box sets the boundary conditions for a periodic zone. It is opened from the Boundary Conditions Task Page (p. 2389). See Inputs for Periodic Boundaries (p. 387) for details about the items below. See Periodic Flows (p. 582) for information about fully-developed periodic flow.



Controls

Zone Name
sets the name of the zone.

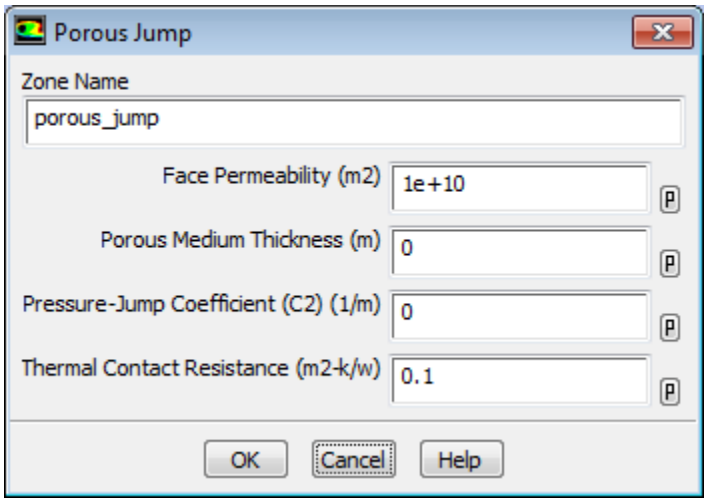
Periodic Type
indicates whether the periodicity of the domain is **Translational** or **Rotational**.

Periodic Pressure Jump
sets the pressure increase/decrease across the periodic boundary. (This item will not appear if the pressure-based (default) solver is used; it is relevant only for the density-based solvers.)

37.8.14. Porous Jump Dialog Box

The **Porous Jump** dialog box sets the boundary conditions for a porous-jump zone. It is opened from the Boundary Conditions Task Page (p. 2389). See Porous Jump Boundary Conditions (p. 403) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone Name
sets the name of the zone.

Face Permeability
sets the face permeability coefficient (α in Equation 6.140 (p. 403)).

Porous Medium Thickness
sets the thickness of the porous medium (Δm).

Pressure-Jump Coefficient
sets the pressure-jump coefficient (C_2).

Thermal Contact Resistance
sets the thermal contact resistance.

Jump Adhesion
sets the adhesion method and contact angle.

Constrained-Two-Sided Adhesion
constrains the contact angle at the porous jump, When this option is disabled then the forced two-sided adhesion treatment is in effect. See Jump Adhesion for more information.

Contact Angle
is the contact angle at the porous jump.

Discrete Phase BC Type
sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

interior
allows the particles to pass through the boundary.

reflect
rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See Particle Reflection at Wall in the *Fluent Theory Guide*.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

trap
terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 23.25: "Trap" Boundary Condition for the Discrete Phase (p. 1303).

escape
reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 23.26: "Escape" Boundary Condition for the Discrete Phase (p. 1303).

wall-jet
indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 16.6: "Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

user-defined
specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function
sets the user-defined function from the drop-down list.

Solar Boundary Conditions
contains the settings for solar ray tracing. This group box is available only if you select **Solar Ray Tracing** from the **Model** list in the **Solar Load** group box of the **Radiation Model** dialog box. See Solar Ray Tracing (p. 900) for details.

Participates in Solar Ray Tracing
specifies whether or not the porous jump participates in solar ray tracing.

Absorptivity
contains the settings that define the absorptivity of the porous jump.

Direct Visible
specifies a multiplier (ranging from 0 to 1) that is applied to the visible portion of the direct solar radiation spectrum to account for the absorption of the porous jump.

Direct IR
specifies a multiplier (ranging from 0 to 1) that is applied to the infrared portion of the direct solar radiation spectrum to account for the absorption of the porous jump.

Transmissivity
contains the settings that define the transmissivity of the porous jump.

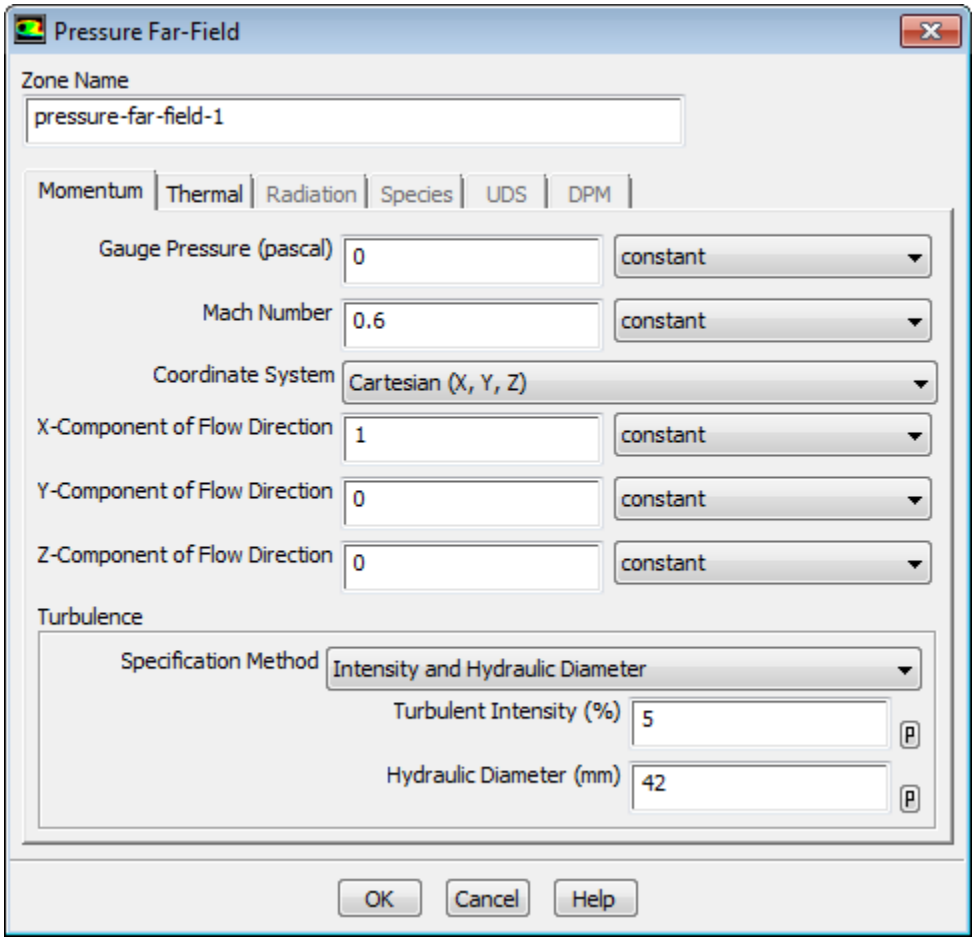
Direct Visible
specifies a multiplier (ranging from 0 to 1) that is applied to the visible portion of the direct solar radiation spectrum to account for the transmissivity of the porous jump.

Direct IR
specifies a multiplier (ranging from 0 to 1) that is applied to the infrared portion of the direct solar radiation spectrum to account for the transmissivity of the porous jump.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.8.15. Pressure Far-Field Dialog Box

The **Pressure Far-Field** dialog box sets the boundary conditions for a pressure far-field zone. It is opened from the Boundary Conditions Task Page (p. 2389). See Inputs at Pressure Far-Field Boundaries (p. 347) for details about defining the items below.



Controls

Zone Name
sets the name of the zone.

Momentum
contains the momentum parameters.

Gauge Pressure
sets the far-field gauge static pressure.

Mach Number
sets the far-field Mach number. The Mach number can be subsonic, sonic, or supersonic.

Coordinate System
allows you to select a **Cartesian**, **Cylindrical**, or **Local Cylindrical** coordinate system. This option is available only for 3D geometry.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X-, Y-, Z-Component of Flow Direction
set the far-field flow direction. These items will appear if the selected **Coordinate System** is **Cartesian** or the model is 2D non-axisymmetric.

Radial-, Tangential-, Axial-Component of Flow Direction
set the far-field flow direction. These items will appear for 2D axisymmetric cases, or for 3D cases for which the selected **Coordinate System** is **Cylindrical** or **Local Cylindrical**.

Axis Origin
sets the **X**, **Y**, and **Z** coordinates of the origin of the local cylindrical coordinate system.

Axis Direction
sets the **X**, **Y**, and **Z** components of the direction of the local cylindrical coordinate system.

Turbulence
contains the turbulence parameters.

Specification Method
specifies which method will be used to define the turbulence parameters. You can choose **K and Epsilon** (k- ε models and RSM only), **K and Omega** (k- ω models only), **Intensity and Length Scale**, **Intensity and Viscosity Ratio**, **Intensity and Hydraulic Diameter**, **Modified Turbulent Viscosity** (Spalart-Allmaras model only), or **Turbulent Viscosity Ratio** (Spalart-Allmaras model only). See Determining Turbulence Parameters (p. 303) for information about the inputs for each of these methods. (This item will appear only for turbulent flow calculations.)

Turbulent Kinetic Energy, Turbulent Dissipation Rate
set values for the turbulence kinetic energy k and its dissipation rate ε. These items will appear if you choose **K and Epsilon** as the **Specification Method**.

Turbulent Kinetic Energy, Specific Dissipation Rate
set values for the turbulence kinetic energy k and its specific dissipation rate ω. These items will appear if you choose **K and Omega** as the **Specification Method**.

Turbulent Intensity, Turbulent Length Scale
set values for turbulence intensity I and turbulence length scale ℓ. These items will appear if you choose **Intensity and Length Scale** as the **Specification Method**.

Turbulent Intensity, Turbulent Viscosity Ratio
set values for turbulence intensity I and turbulent viscosity ratio μ_t / μ. These items will appear if you choose **Intensity and Viscosity Ratio** as the **Specification Method**.

Turbulent Intensity, Hydraulic Diameter
set values for turbulence intensity I and hydraulic diameter L. These items will appear if you choose **Intensity and Hydraulic Diameter** as the **Specification Method**.

Modified Turbulent Viscosity
sets the value of the modified turbulent viscosity $\tilde{\nu}$. This item will appear if you choose **Modified Turbulent Viscosity** as the **Specification Method**.

Turbulent Viscosity Ratio
sets the value of the turbulent viscosity ratio μ_t / μ. This item will appear if you choose **Turbulent Viscosity Ratio** as the **Turbulence Specification Method**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reynolds-Stress Specification Method
specifies which method will be used to determine the Reynolds stress boundary conditions when the Reynolds stress turbulence model is used. You can choose either **K or Turbulent Intensity** or **Reynolds-Stress Components**. If you choose the former, ANSYS Fluent will compute the Reynolds stresses for you. If you choose the latter, you will explicitly specify the Reynolds stresses yourself. See Reynolds Stress Model (p. 808) for details. (This item will appear only for RSM turbulent flow calculations.)

UU, VV, WW, UV, VW, UW Reynolds Stresses
specify the Reynolds stress components when **Reynolds-Stress Components** is chosen as the **Reynolds-Stress Specification Method**.

Thermal
contains the thermal parameters.

Temperature
sets the far-field static temperature.

Radiation
contains the boundary conditions for the radiation model at the pressure far-field zone.

External Black Body Temperature Method, Internal Emissivity
set the radiation boundary conditions when you are using the P-1, DTRM, DO, S2S, or MC models for radiation heat transfer. See Defining Boundary Conditions for Radiation (p. 869) for details.

Participates in Solar Ray Tracing
specifies whether or not the pressure far-field zone participates in solar ray tracing.

Solar Transmissivity Factor
specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the pressure far-field zone.

Participates in View Factor Calculation
specifies whether or not the pressure far-field zone participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

Species
contains the species parameters.

Specify Species in Mole Fractions
allows you to specify the species in mole fractions rather than mass fractions.

Species Mass Fractions
contains inputs for the mass fractions of defined species. See Defining Cell Zone and Boundary Conditions for Species (p. 990) for details about these inputs. (These items will appear only if you are modeling non-reacting multi-species flow or you are using the finite-rate reaction formulation.)

Mean Mixture Fraction, Mixture Fraction Variance
set inlet values for the PDF mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model.)

Secondary Mean Mixture Fraction, Secondary Mixture Fraction Variance
set inlet values for the secondary mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model with two mixture fractions.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Progress Variable
sets the value of the progress variable for premixed turbulent combustion. See Setting Boundary Conditions for the Progress Variable (p. 1094) for details.

This item will appear only if the premixed or partially premixed combustion model is used.

UDS
contains the UDS parameters.

User-Defined Scalar Boundary Condition
appears only if user-defined scalars are specified.

User Scalar n
specifies whether the scalar is a specified flux or a specified value.

User-Defined Scalar Boundary Value
appears only if user-defined scalars are specified.

User Scalar n
specifies the value of the scalar.

DPM
contains the discrete phase parameters.

Discrete Phase BC Type
sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect
rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See Particle Reflection at Wall in the *Fluent Theory Guide*.)

trap
terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 23.25: "Trap" Boundary Condition for the Discrete Phase (p. 1303).

escape
reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 23.26: "Escape" Boundary Condition for the Discrete Phase (p. 1303).

wall-jet
indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 16.6: "Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

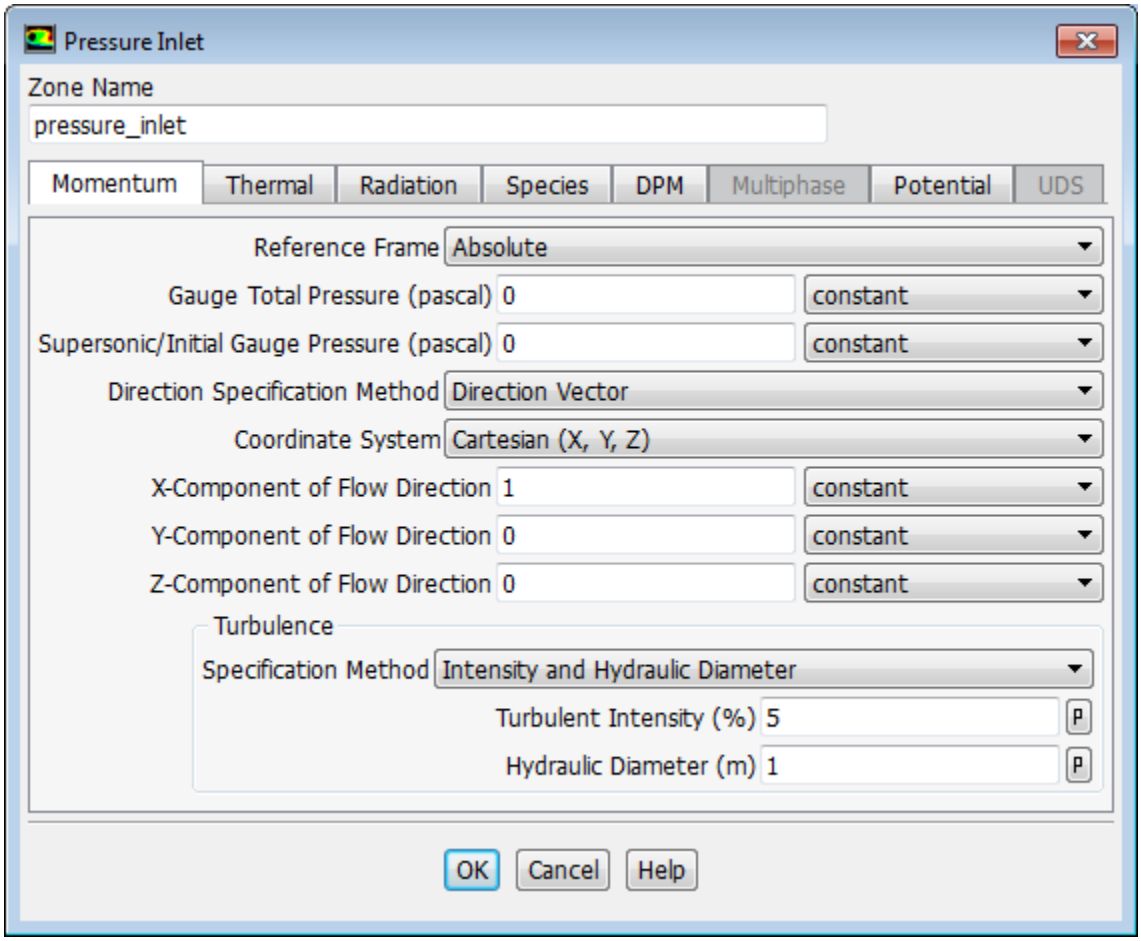
user-defined
specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function
sets the user-defined function from the drop-down list.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.8.16. Pressure Inlet Dialog Box

The **Pressure Inlet** dialog box sets the boundary conditions for a pressure inlet zone. It is opened from the Boundary Conditions Task Page (p. 2389). See Inputs at Pressure Inlet Boundaries (p. 308) for details about defining the items below.



Controls

Zone Name
sets the name of the zone.

Momentum
contains the momentum parameters.

Reference Frame
specifies the reference frame for the pressure inlet. If the cell zone adjacent to the pressure inlet is moving, you can choose to specify the total temperature, total pressure, and velocity components as **Relative to Adjacent CellZone** or **Absolute** in the **Reference Frame** drop-down list.

Gauge Total Pressure
sets the gauge total (or stagnation) pressure of the inflow stream. If you are using moving reference frames, see Defining Total Pressure and Temperature (p. 310) for information about relative and absolute total pressure.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Supersonic/Initial Gauge Pressure
sets the static pressure on the boundary when the flow becomes (locally) supersonic. It is also used to compute initial values for pressure, temperature, and velocity if the pressure inlet boundary condition is selected for computing initial values (see Initializing the Entire Flow Field Using Standard Initialization (p. 1591)).

Direction Specification Method
specifies the method you will use to define the flow direction. If you choose **Direction Vector**, you will define the flow direction components, and if you choose **Normal to Boundary** no inputs are required. See Defining the Flow Direction (p. 311) for information on specifying flow direction.

Coordinate System
specifies whether **Cartesian**, **Cylindrical**, **Local Cylindrical**,or **Local Cylindrical Swirl** vector components will be defined. This item will appear only for 3D cases in which you have selected **Direction Vector** as the **Direction Specification Method**.

X,Y,Z-Component of Flow Direction
set the direction of the flow at the inlet boundary. These items will appear if the selected **Coordinate System** is **Cartesian** or the model is 2D non-axisymmetric.

Radial, Tangential, Axial Component of Flow Direction
set the direction of the flow at the inlet boundary. These items will appear for 2D axisymmetric cases, or for 3D cases for which the selected **Coordinate System** is **Cylindrical** or **Local Cylindrical**.

Radial-, Axial-Component of Flow Direction, Tangential-Velocity
appear for a 3D **Local Cylindrical Swirl** coordinate system.

Axis Origin
sets the **X**, **Y**, and **Z** coordinates of the origin of the local cylindrical (swirl) coordinate system.

Axis Direction
sets the **X**, **Y**, and **Z** components of the direction of the local cylindrical (swirl) coordinate system.

Turbulence
contains the turbulence parameters.

Specification Method
specifies which method will be used to define the turbulence parameters. You can choose **K and Epsilon** (k- ε models and RSM only), **K and Omega** (k- ω models only), **Intensity and Length Scale**, **Intensity and Viscosity Ratio**, **Intensity and Hydraulic Diameter**, **Modified Turbulent Viscosity** (Spalart-Allmaras model only), or **Turbulent Viscosity Ratio** (Spalart-Allmaras model only). See Determining Turbulence Parameters (p. 303) for information about the inputs for each of these methods. (This item will appear only for turbulent flow calculations.)

Turbulent Kinetic Energy, Turbulent Dissipation Rate
set values for the turbulence kinetic energy k and its dissipation rate ε. These items will appear if you choose **K and Epsilon** as the **Specification Method**.

Turbulent Kinetic Energy, Specific Dissipation Rate
set values for the turbulence kinetic energy k and its specific dissipation rate ω. These items will appear if you choose **K and Omega** as the **Specification Method**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Turbulent Intensity, Turbulent Length Scale
set values for turbulence intensity I and turbulence length scale ℓ. These items will appear if you choose **Intensity and Length Scale** as the **Specification Method**.

Turbulent Intensity, Turbulent Viscosity Ratio
set values for turbulence intensity I and turbulent viscosity ratio μ_t / μ. These items will appear if you choose **Intensity and Viscosity Ratio** as the **Specification Method**.

Turbulent Intensity, Hydraulic Diameter
set values for turbulence intensity I and hydraulic diameter L. These items will appear if you choose **Intensity and Hydraulic Diameter** as the **Specification Method**.

Modified Turbulent Viscosity
sets the value of the modified turbulent viscosity $\tilde{\nu}$. This item will appear if you choose **Modified Turbulent Viscosity** as the **Specification Method**.

Turbulent Viscosity Ratio
sets the value of the turbulent viscosity ratio μ_t / μ. This item will appear if you choose **Turbulent Viscosity Ratio** as the **Specification Method**.

Reynolds-Stress Specification Method
specifies which method will be used to determine the Reynolds stress boundary conditions when the Reynolds stress turbulence model is used. You can choose either **K or Turbulent Intensity** or **Reynolds-Stress Components**. If you choose the former, ANSYS Fluent will compute the Reynolds stresses for you. If you choose the latter, you will explicitly specify the Reynolds stresses yourself. See Reynolds Stress Model (p. 808) for details. (This item will appear only for RSM turbulent flow calculations.)

UU, VV, WW, UV, VW, UW Reynolds Stresses
specify the Reynolds stress components when **Reynolds-Stress Components** is chosen as the **Reynolds-Stress Specification Method**.

Acoustic Wave Model
contains settings for treatment of acoustic pressure waves at the boundary.

Off
disables special treatment of acoustic pressure waves at the boundary.

Non Reflecting
enables the general non-reflecting boundary condition treatment described in General Non-Reflecting Boundary Conditions (p. 416).

Impedance
enables the impedance boundary condition treatment described in Impedance Boundary Conditions (p. 422).

Transparent Flow Forcing
enables the transparent flow forcing boundary condition described in Transparent Flow Forcing Boundary Conditions (p. 425).

Impedance Parameters
contains the parameters for the impedance boundary condition treatment. For details refer to Using the Impedance Boundary Condition (p. 424).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Transparent Flow Forcing Parameters
contains the parameters for the transparent flow forcing boundary condition treatment. For details refer to Using the Transparent Flow Forcing Boundary Condition (p. 426).

Thermal
contains the thermal parameters.

Total Temperature
sets the total temperature of the inflow stream. If you are using moving reference frames, see Defining Total Pressure and Temperature (p. 310) for information about relative and absolute total temperature.

Radiation
contains the boundary conditions for the radiation model at the pressure inlet.

External Black Body Temperature Method, Internal Emissivity
set the radiation boundary conditions when you are using the P-1, DTRM, DO, S2S, or MC models for radiation heat transfer. See Defining Boundary Conditions for Radiation (p. 869) for details.

Participates in Solar Ray Tracing
specifies whether or not the pressure inlet participates in solar ray tracing.

Solar Transmissivity Factor
specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the pressure inlet.

Participates in View Factor Calculation
specifies whether or not the pressure inlet participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

Species
contains the species parameters.

Specify Species in Mole Fractions
allows you to specify the species in mole fractions rather than mass fractions.

Species Mass Fractions
contains inputs for the mass fractions of defined species. See Defining Cell Zone and Boundary Conditions for Species (p. 990) for details about these inputs. (These items will appear only if you are modeling non-reacting multi-species flow or you are using the finite-rate reaction formulation.)

Mean Mixture Fraction, Mixture Fraction Variance
set inlet values for the PDF mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model.)

Secondary Mean Mixture Fraction, Secondary Mixture Fraction Variance
set inlet values for the secondary mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model with two mixture fractions.)

Progress Variable
sets the value of the progress variable for premixed turbulent combustion. See Setting Boundary Conditions for the Progress Variable (p. 1094) for details.

This item will appear only if the premixed or partially premixed combustion model is used.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DPM

contains the discrete phase parameters.

Discrete Phase BC Type

sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect

rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See Particle Reflection at Wall in the *Fluent Theory Guide*.)

trap

terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 23.25: "Trap" Boundary Condition for the Discrete Phase (p. 1303).

escape

reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 23.26: "Escape" Boundary Condition for the Discrete Phase (p. 1303).

wall-jet

indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 16.6: "Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

user-defined

specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function

sets the user-defined function from the drop-down list.

Multiphase

contains the multiphase parameters.

Granular Temperature

specifies temperature for the solids phase and is proportional to the kinetic energy of the random motion of the particles.

Volume Fraction

specifies the volume fraction of the secondary phase selected in the Boundary Conditions Task Page (p. 2389). This section of the dialog box will appear when one of the multiphase models is being used. See Defining Multiphase Cell Zone and Boundary Conditions (p. 1391) for details.

Potential

displays the boundary conditions for the electric potential field. This tab is available only if you have enabled either the **Electric Potential model** or the **Electrochemical** reaction model in the Species Dialog Box (p. 2332).

Potential Boundary Condition

is a drop-down list of available potential boundary condition types: **Specified Flux** and **Specified Value**. For the **Specified Flux** boundary condition, you will need to specify **Current Density** at the wall. For the **Specified Value** boundary condition, you will need to specify **Potential** at the wall.

UDS

contains the UDS parameters.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

User-Defined Scalar Boundary Condition
appears only if user-defined scalars are specified.

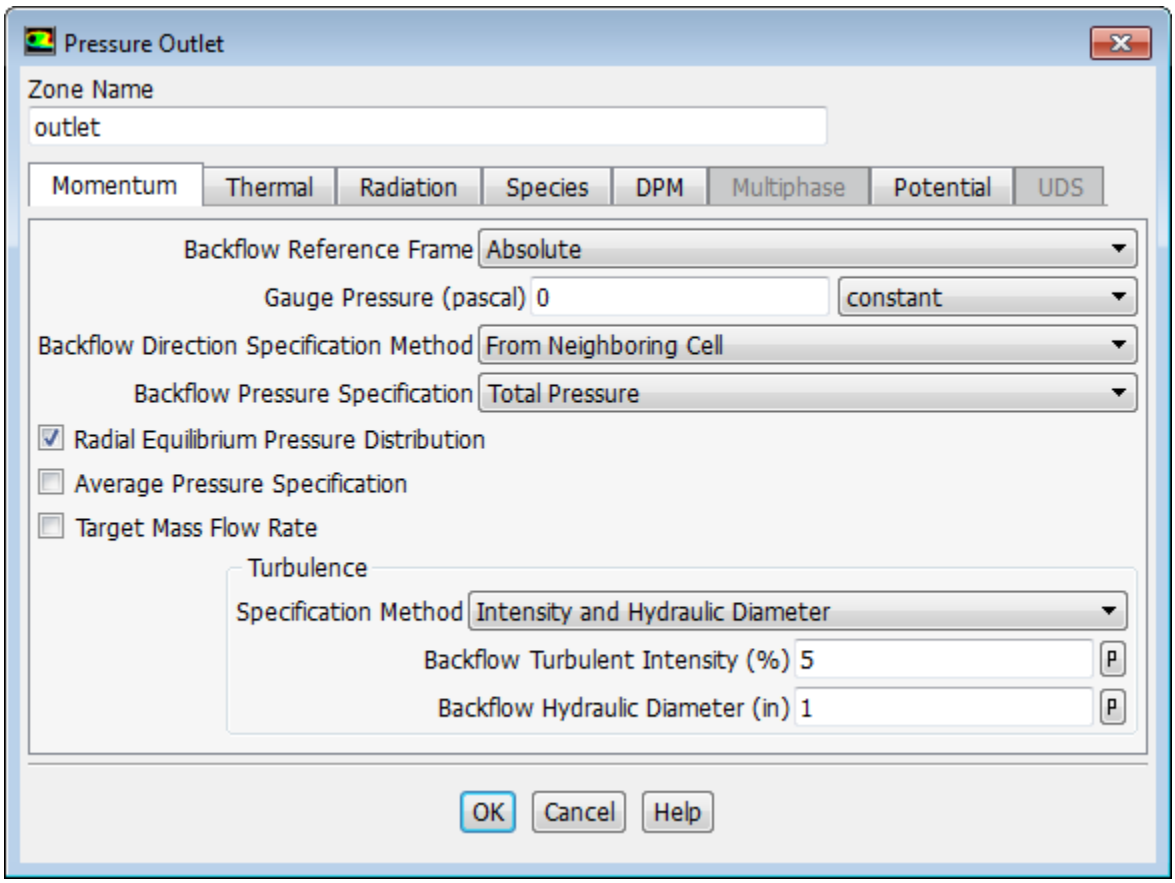
User Scalar n
specifies whether the scalar is a specified flux or a specified value.

User-Defined Scalar Boundary Value
appears only if user-defined scalars are specified.

User Scalar n
specifies the value of the scalar.

37.8.17. Pressure Outlet Dialog Box

The **Pressure Outlet** dialog box sets the boundary conditions for a pressure outlet zone. It is opened from the Boundary Conditions Task Page (p. 2389). See Inputs at Pressure Outlet Boundaries (p. 338) for details about defining the items below.



Controls

Zone Name
sets the name of the zone.

Momentum
contains the momentum parameters.

Backflow Reference Frame
specify whether backflow temperature, pressure, and flow directions are in the **Absolute** or **Relative to the Adjacent Cell Zone** reference frame.

Gauge Pressure
sets the gauge pressure at the outflow boundary.

Backflow Direction Specification Method
sets the direction of the inflow stream should the flow reverse direction. If you choose **Direction Vector**, you will define the flow direction components, and if you choose **Normal to Boundary** or **From Neighboring Cell**, no inputs are required. See Inputs at Pressure Outlet Boundaries (p. 338) for information on specifying flow direction.

Coordinate System
contains a drop-down list for selecting the coordinate system. You can choose **Cartesian**, **Cylindrical**, or **Local Cylindrical**. This option is available only when **Direction Vector** is selected from the **Backflow Direction Specification Method** drop-down list.

X-, Y-, Z-Component of Flow Direction
allows you to specify the velocity components in x, y, and z directions respectively. This option is available when **Cartesian** is selected for the **Coordinate System**.

Radial-, Tangential-, Axial-Component of Flow Direction
set the direction of the flow at the boundary. These items will appear for 2D axisymmetric cases, or for 3D cases for which the selected **Coordinate System** is **Cylindrical** or **Local Cylindrical**.

Backflow Pressure Specification
specifies how the pressure is calculated under backflow conditions. When the NRBC option is not enabled, if you select **Static Pressure**, the **Gauge Pressure** is directly imposed as the boundary face pressure; if you select **Total Pressure**, the **Gauge Pressure** will be combined with a dynamic contribution that is based on the velocity in the adjacent cell zone. For NRBCs, if you select **Static Pressure**, the **Gauge Pressure** is directly imposed as the pressure at infinity; if you select **Total Pressure**, the pressure at infinity is a calculated value that is based on the **Gauge Pressure**.

Radial Equilibrium Pressure Distribution
enables the radial equilibrium pressure distribution. See Defining Static Pressure (p. 339) for details.

This item appears only for 3D and axisymmetric swirl solvers.

Average Pressure Specification
allows the pressure along the outlet boundary to vary, but maintain an average equivalent to the specified value in the **Gauge Pressure** input field. In this boundary implementation, the pressure variation provides a low level of non-reflectivity. For more details, see Calculation Procedure at Pressure Outlet Boundaries (p. 342).

Note

The **Average Pressure Specification** option is not available if the **Radial Equilibrium Pressure Distribution** option is enabled.

Target Mass Flow Rate
allows you to set mass flow rate as a boundary condition at the outlet.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Target Mass Flow
allows you to specify the flow as either a constant value or a user-defined function.

Upper Limit of Absolute Pressure, Lower Limit of Absolute Pressure
specifies the range of the pressure limits, which have different pressure variations on different boundaries. The upper and lower pressure limits can be specified as a constant or a profile.

Turbulence
contains the turbulence parameters.

Specification Method
specifies which method will be used to define the turbulence parameters. You can choose **K and Epsilon** (k- ε models and RSM only), **K and Omega** (k- ω models only), **Intensity and Length Scale**, **Intensity and Viscosity Ratio**, **Intensity and Hydraulic Diameter**, **Modified Turbulent Viscosity** (Spalart-Allmaras model only), or **Turbulent Viscosity Ratio** (Spalart-Allmaras model only). See Determining Turbulence Parameters (p. 303) for information about the inputs for each of these methods. (This item will appear only for turbulent flow calculations.)

Backflow Turbulent Kinetic Energy, Backflow Turbulent Dissipation Rate
set values for the turbulence kinetic energy k and its dissipation rate ε. These items will appear if you choose **K and Epsilon** as the **Specification Method**.

Backflow Turbulent Kinetic Energy, Backflow Specific Dissipation Rate
set values for the turbulence kinetic energy k and its specific dissipation rate ω. These items will appear if you choose **K and Omega** as the **Specification Method**.

Backflow Turbulent Intensity, Backflow Turbulent Length Scale
set values for turbulence intensity I and turbulence length scale ℓ. These items will appear if you choose **Intensity and Length Scale** as the **Specification Method**.

Backflow Turbulent Intensity, Backflow Turbulent Viscosity Ratio
set values for turbulence intensity I and turbulent viscosity ratio μ_t / μ. These items will appear if you choose **Intensity and Viscosity Ratio** as the **Specification Method**.

Backflow Turbulent Intensity, Backflow Hydraulic Diameter
set values for turbulence intensity I and hydraulic diameter L. These items will appear if you choose **Intensity and Hydraulic Diameter** as the **Specification Method**.

Backflow Modified Turbulent Viscosity
sets the value of the backflow modified turbulent viscosity $\tilde{\nu}$. This item will appear if you choose **Modified Turbulent Viscosity** as the **Specification Method**.

Backflow Turbulent Viscosity Ratio
sets the value of the backflow turbulent viscosity ratio μ_t / μ. This item will appear if you choose **Turbulent Viscosity Ratio** as the **Specification Method**.

Reynolds-Stress Specification Method
specifies which method will be used to determine the backflow Reynolds stress boundary conditions when the Reynolds stress turbulence model is used. You can choose either **K or Turbulent Intensity** or **Reynolds-Stress Components**. If you choose the former, ANSYS Fluent will compute the Reynolds stresses for you. If you choose the latter, you will explicitly specify the Reynolds stresses yourself. See Reynolds Stress Model (p. 808) for details. (This item will appear only for RSM turbulent flow calculations.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Backflow UU, VV, WW, UV, VW, UW Reynolds Stresses
specify the backflow Reynolds stress components when **Reynolds-Stress Components** is chosen as the **Reynolds-Stress Specification Method**.

Acoustic Wave Model
contains settings for treatment of acoustic pressure waves at the boundary.

Off
disables special treatment of acoustic pressure waves at the boundary.

Non Reflecting
enables the general non-reflecting boundary condition treatment described in General Non-Reflecting Boundary Conditions (p. 416).

Impedance
enables the impedance boundary condition treatment described in Impedance Boundary Conditions (p. 422).

Transparent Flow Forcing
enables the transparent flow forcing boundary condition described in Transparent Flow Forcing Boundary Conditions (p. 425).

Non Reflecting Parameters
contains settings for the non-reflecting boundary condition treatment.

Exit Pressure Specification
defines how the pressure is calculated. For **Pressure at Infinity**, the pressure at the boundary relaxes toward the **Gauge Pressure** at infinity; for **Average Boundary Pressure**, an effort is made to force the average pressure on the boundary to approach the **Gauge Pressure** value.

Impedance Parameters
contains the parameters for the impedance boundary condition treatment. For details refer to Using the Impedance Boundary Condition (p. 424).

Transparent Flow Forcing Parameters
contains the parameters for the transparent flow forcing boundary condition treatment. For details refer to Using the Transparent Flow Forcing Boundary Condition (p. 426).

Thermal
contains the thermal parameters.

Backflow Total Temperature
sets the total temperature of the inflow stream should the flow reverse direction

Radiation
contains the boundary conditions for the radiation model at the pressure outlet.

External Black Body Temperature Method, Internal Emissivity
set the radiation boundary conditions when you are using the P-1, DTRM, DO, S2S, or MC models for radiation heat transfer. See Defining Boundary Conditions for Radiation (p. 869) for details.

Participates in Solar Ray Tracing
specifies whether or not the pressure outlet participates in solar ray tracing.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solar Transmissivity Factor
specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the pressure outlet.

Participates in View Factor Calculation
specifies whether or not the pressure outlet participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

Species
contains the species parameters.

Specify Species in Mole Fractions
allows you to specify the species in mole fractions rather than mass fractions.

Mean Mixture Fraction, Mixture Fraction Variance
set inlet values for the PDF mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model.)

Secondary Mean Mixture Fraction, Secondary Mixture Fraction Variance
set inlet values for the secondary mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model with two mixture fractions.)

Species Mass Fractions
contains inputs for the mass fractions of defined species. See Defining Cell Zone and Boundary Conditions for Species (p. 990) for details about these inputs. These items will appear only if you are modeling non-reacting multi-species flow or you are using the finite-rate reaction formulation.

Backflow Progress Variable
sets the value of the progress variable for premixed turbulent combustion. See Setting Boundary Conditions for the Progress Variable (p. 1094) for details.

This item will appear only if the premixed or partially premixed combustion model is used.

DPM
contains the discrete phase parameters.

Discrete Phase BC Type
sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect
rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See Particle Reflection at Wall in the *Fluent Theory Guide*.)

trap
terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 23.25: "Trap" Boundary Condition for the Discrete Phase (p. 1303).

escape
reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 23.26: "Escape" Boundary Condition for the Discrete Phase (p. 1303).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

wall-jet
indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 16.6:"Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

user-defined
specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function
sets the user-defined function from the drop-down list.

Multiphase
contains the multiphase parameters.

Backflow Granular Temperature
specifies temperature for the solids phase and is proportional to the kinetic energy of the random motion of the particles.

Volume Fraction Specification Method
sets the method used to specify the volume fraction of the secondary phase selected in the Boundary Conditions Task Page (p. 2389). This section of the dialog box will appear when one of the multiphase models is being used. See Defining Multiphase Cell Zone and Boundary Conditions (p. 1391) for details.

Backflow Volume Fraction
specifies the volume fraction of the secondary phase as a constant, profile, of UDF function.

From Neighboring Cell
calculates the volume fraction from the neighboring cells.

Potential
displays the boundary conditions for the electric potential field. This tab is available only if you have enabled either the **Electric Potential model** or the **Electrochemical** reaction model in the Species Dialog Box (p. 2332).

Potential Boundary Condition
is a drop-down list of available potential boundary condition types: **Specified Flux** and **Specified Value**. For the **Specified Flux** boundary condition, you will need to specify **Current Density** at the wall. For the **Specified Value** boundary condition, you will need to specify **Potential** at the wall.

UDS
contains the UDS parameters.

User-Defined Scalar Boundary Condition
appears only if user-defined scalars are specified.

User Scalar n
specifies whether the scalar is a specified flux or a specified value.

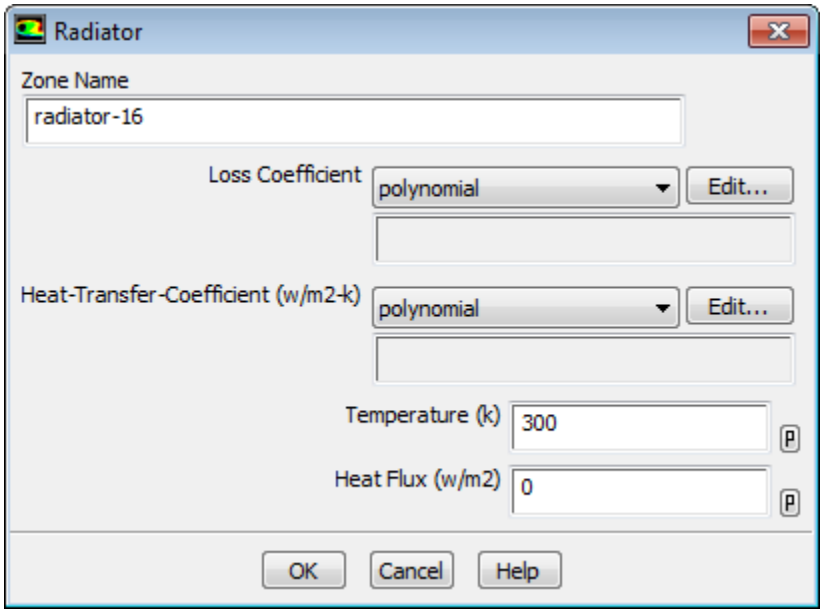
User-Defined Scalar Boundary Value
appears only if user-defined scalars are specified.

User Scalar n
specifies the value of the scalar.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.8.18. Radiator Dialog Box

The **Radiator** dialog box sets the boundary conditions for a radiator model zone. It is opened from the Boundary Conditions Task Page (p. 2389). See User Inputs for Radiators (p. 398) for details about the items below.



Controls

Zone Name
sets the name of the zone.

Loss Coefficient
specifies the loss coefficient as a constant value or as a polynomial, piecewise-linear, or piecewise-polynomial function of velocity. See Defining the Pressure Loss Coefficient Function (p. 399) for details.

Heat-Transfer-Coefficient
specifies the heat-transfer coefficient as a constant value or as a polynomial, piecewise-linear, or piecewise-polynomial function of velocity. See Defining the Heat Flux Parameters (p. 401) for details.

Temperature
sets the temperature used to compute heat flux from the radiator using the **Heat-Transfer-Coefficient**. If **Temperature** is absolute zero, the **Heat Flux** condition is used instead.

Heat Flux
sets the heat flux at the radiator surface (used only when **Temperature** is absolute zero).

Discrete Phase BC Type
sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

interior
allows the particles to pass through the boundary.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

reflect
rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See Particle Reflection at Wall in the *Fluent Theory Guide*.)

trap
terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 23.25: "Trap" Boundary Condition for the Discrete Phase (p. 1303).

escape
reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 23.26: "Escape" Boundary Condition for the Discrete Phase (p. 1303).

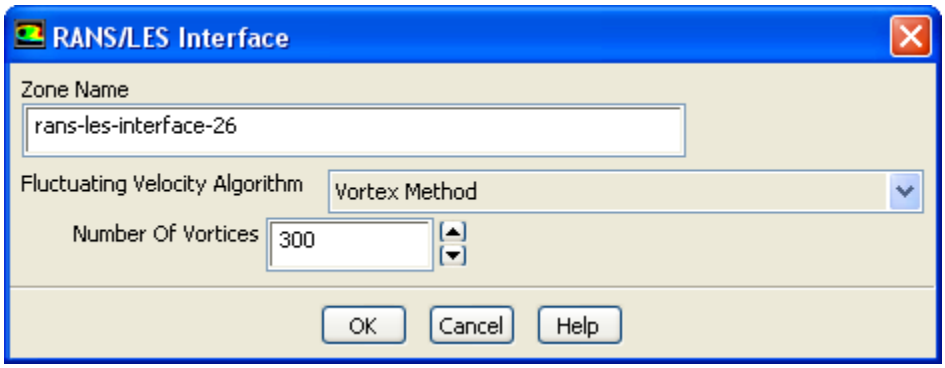
wall-jet
indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 16.6: "Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

user-defined
specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function
sets the user-defined function from the drop-down list.

37.8.19. RANS/LES Interface Dialog Box

The **RANS/LES Interface** dialog box can be used to create artificial resolved turbulence (fluctuation/perturbations) at the interface where the flow proceeds from the RANS zone into the LES zone of the computational domain for Embedded LES turbulent flows. It is opened from the Boundary Conditions Task Page (p. 2389). See Setting Up the Embedded Large Eddy Simulation (ELES) Model (p. 793) for more information about RANS/LES interfaces.



Controls

Zone Name
sets the name of the RANS/LES interface.

Fluctuating Velocity Algorithm
displays the methods for generating fluctuating velocity components at the RANS/LES interface. Available options include:

- No Perturbations

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Spectral Synthesizer
- Vortex Method

Number of Vortices

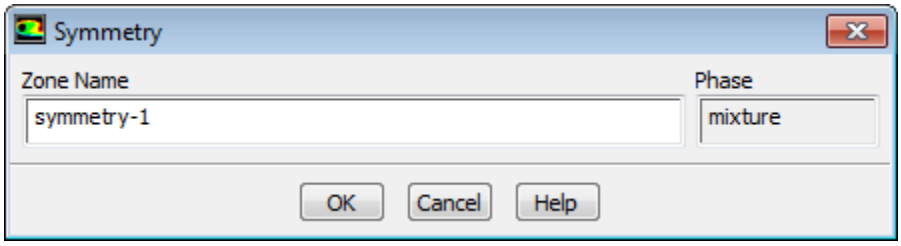
displays the amount of vortices that the selected fluctuating velocity method distributes randomly over the face zone and uses to generate turbulent fluctuations (available for the Vortex Method only).

Important

The RANS/LES 'interface' can either be an interior zone (that has been assigned to be a **rans-les-interface** zone), or it can be a non-conformal interface. If it is a non-conformal interface, then you need to identify the name of the non-conformal interface's "interior" zone (for example, using the Mesh Interfaces Dialog Box (p. 2732)), and then go to that zone in the **Boundary Conditions** task page.

37.8.20. Symmetry Dialog Box

The **Symmetry** dialog box can be used to modify the name of a symmetry zone; there are no conditions to be set. It is opened from the Boundary Conditions Task Page (p. 2389). See Symmetry Boundary Conditions (p. 384) for information about symmetry boundaries.



Controls

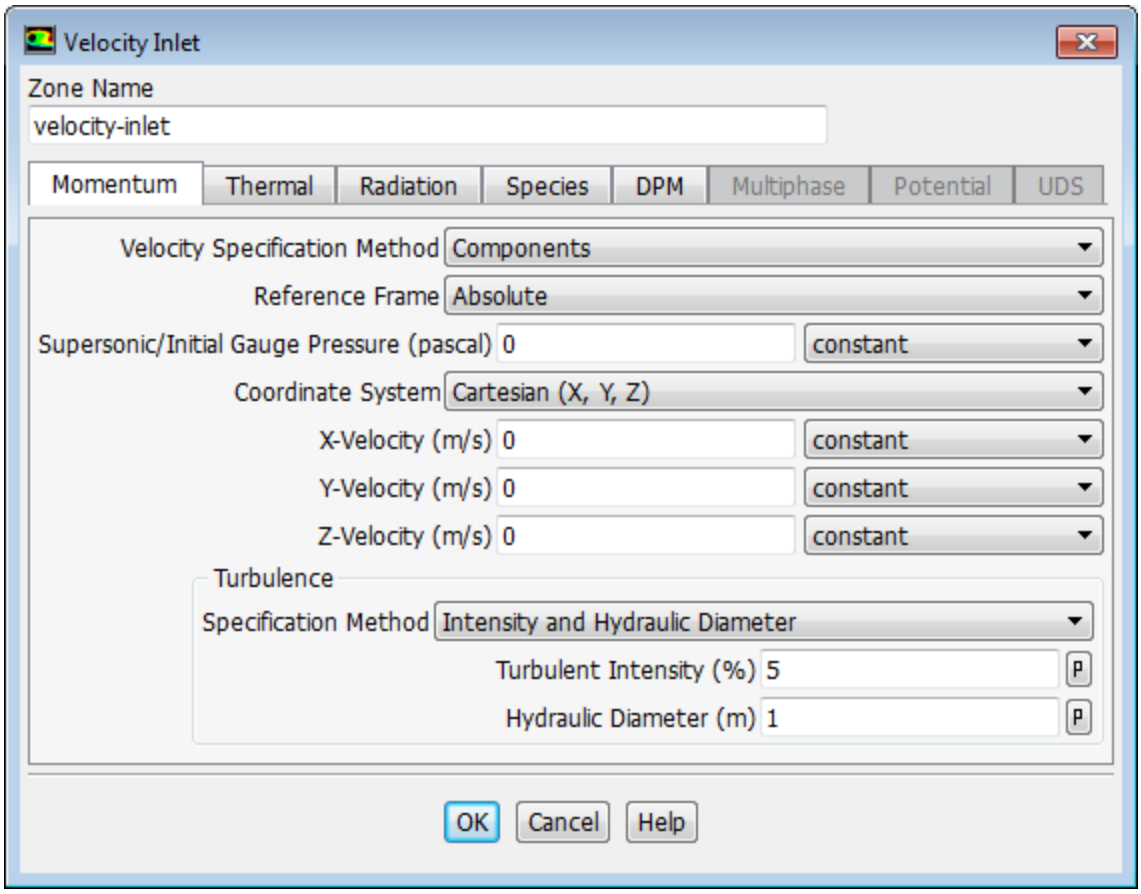
Zone Name
sets the name of the zone.

Phase
displays the name of the phase. This item is available only for multiphase flows.

37.8.21. Velocity Inlet Dialog Box

The **Velocity Inlet** dialog box sets the boundary conditions for a velocity inlet zone. It is opened from the Boundary Conditions Task Page (p. 2389). See Inputs at Velocity Inlet Boundaries (p. 317) for details about defining the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone Name
sets the name of the zone.

Phase
displays the name of the phase. This item appears if the VOF, mixture, or Eulerian multiphase model is being used.

Open Channel Wave BC
allows you to set specific parameters for a particular boundary for open channel wave boundaries. This is available when the volume of fluid multiphase model is selected.

Momentum
contains the momentum parameters.

Velocity Specification Method
sets the method used to define the inflow velocity.

Flow Direction Specification Method
sets the method used to define the direction of flow of the wave. This is available when you enable the **Open Channel Wave BC** option.

Magnitude and Direction
allows specification in terms of a **Velocity Magnitude** and **Flow-Direction**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Components
allows specification in terms of the Cartesian, cylindrical, or local cylindrical velocity components.

Magnitude, Normal to Boundary
allows specification of a **Velocity Magnitude** normal to the boundary.

Reference Frame
specifies relative or absolute velocity inputs. You can choose to enter **Absolute** velocities or velocities **Relative to Adjacent Cell Zone**. If you are not using moving reference frames, both options are equivalent, so you need not choose.

Uniform Flow Velocity Magnitude
is the flow velocity, specified as a constant or a parameter.

Coordinate System
specifies whether **Cartesian**, **Cylindrical**, or **Local Cylindrical** velocities will be defined. This item will appear only for 3D cases in which you have selected **Magnitude and Direction** or **Components** as the **Velocity Specification Method**.

X-, Y-, Z-Velocity
set the components of the velocity vector at the inflow boundary. These items will appear for 2D non-axisymmetric models, or for 3D models if you select the **Components** option as the **Velocity Specification Method** and **Cartesian** as the **Coordinate System**.

Radial-, Tangential-, Axial-Velocity
set the components of the velocity vector at the inflow boundary. These items will appear for 3D models if you select the **Components** option as the **Velocity Specification Method** and **Cylindrical** or **Local Cylindrical** as the **Coordinate System**.

Axial-, Radial-, Swirl-Velocity
set the components of the velocity vector at the inflow boundary. These items will appear for 2D axisymmetric models.

Important

Swirl-Velocity will appear only for 2D axisymmetric swirl models.

Angular Velocity
specifies the angular velocity Ω for a 3D flow. This item will appear for a 3D model if you select the **Components** option as the **Velocity Specification Method** and **Cylindrical** or **Local Cylindrical** as the **Coordinate System**.

Swirl Angular Velocity
specifies the swirl angular velocity Ω for an axisymmetric swirling flow. This item will appear for an axisymmetric swirl model if you choose **Components** as the **Velocity Specification Method**.

Velocity Magnitude
sets the magnitude of the velocity vector at the inflow boundary. This item will appear if you select the **Magnitude and Direction** or **Magnitude, Normal to Boundary** option as the **Velocity Specification Method**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X-, Y-, Z-Component of Flow Direction
set the direction of the velocity vector at the inflow boundary. These items will appear for 2D non-axisymmetric models if you select the **Magnitude and Direction** option as the **Velocity Specification Method**, or for 3D models if you select the **Magnitude and Direction** option as the **Velocity Specification Method** and **Cartesian** as the **Coordinate System**.

Radial-, Tangential-, Axial-Component of Flow Direction
set the direction of the velocity vector at the inlet boundary. These items will appear for 3D models if you select the **Magnitude and Direction** option as the **Velocity Specification Method** and **Cylindrical** or **Local Cylindrical** as the **Coordinate System**, or for 2D axisymmetric models.

Important

Tangential-Velocity will appear only for 2D axisymmetric swirl models.

Axis Origin
sets the **X**, **Y**, and **Z** coordinates of the origin of the local cylindrical coordinate system.

Axis Direction
sets the **X**, **Y**, and **Z** components of the direction of the local cylindrical coordinate system.

Outflow Gauge Pressure
specifies the pressure to be used as the pressure outlet condition if flow exits the domain at any face on the velocity inlet boundary. (Note that this effect is similar to that of the "velocity far-field" boundary that was available in RAMPANT 3.)

This item appears only for the density-based solvers.

Turbulence
contains the turbulence parameters.

Specification Method
specifies which method will be used to define the turbulence parameters. You can choose **K and Epsilon** (k- ε models and RSM only), **K and Omega** (k- ω models only), **Intensity and Length Scale**, **Intensity and Viscosity Ratio**, **Intensity and Hydraulic Diameter**, **Modified Turbulent Viscosity** (Spalart-Allmaras model only), or **Turbulent Viscosity Ratio** (Spalart-Allmaras model only). See Determining Turbulence Parameters (p. 303) for information about the inputs for each of these methods. (This item will appear only for turbulent flow calculations.)

Turbulent Kinetic Energy, Turbulent Dissipation Rate
set values for the turbulence kinetic energy k and its dissipation rate ε. These items will appear if you choose **K and Epsilon** as the **Specification Method**.

Turbulent Kinetic Energy, Specific Dissipation Rate
set values for the turbulence kinetic energy k and its specific dissipation rate ω. These items will appear if you choose **K and Omega** as the **Specification Method**.

Turbulent Intensity, Turbulent Length Scale
set values for turbulence intensity I and turbulence length scale ℓ. These items will appear if you choose **Intensity and Length Scale** as the **Specification Method**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Turbulent Intensity, Turbulent Viscosity Ratio

set values for turbulence intensity I and turbulent viscosity ratio μ_t / μ. These items will appear if you choose **Intensity and Viscosity Ratio** as the **Specification Method**.

Turbulent Intensity, Hydraulic Diameter

set values for turbulence intensity I and hydraulic diameter L. These items will appear if you choose **Intensity and Hydraulic Diameter** as the **Specification Method**.

Modified Turbulent Viscosity

sets the value of the modified turbulent viscosity $\tilde{\nu}$. This item will appear if you choose **Modified Turbulent Viscosity** as the **Specification Method**.

Turbulent Viscosity Ratio

sets the value of the turbulent viscosity ratio μ_t / μ. This item will appear if you choose **Turbulent Viscosity Ratio** as the **Specification Method**.

Turbulent Intensity

sets the value of the turbulence intensity I for the LES model.

Reynolds-Stress Specification Method

specifies which method will be used to determine the Reynolds stress boundary conditions when the Reynolds stress turbulence model is used. You can choose either **K or Turbulent Intensity** or **Reynolds-Stress Components**. If you choose the former, ANSYS Fluent will compute the Reynolds stresses for you. If you choose the latter, you will explicitly specify the Reynolds stresses yourself. See Reynolds Stress Model (p. 808) for details. (This item will appear only for RSM turbulent flow calculations.)

UU, VV, WW, UV, VW, UW Reynolds Stresses

specify the Reynolds stress components when **Reynolds-Stress Components** is chosen as the **Reynolds-Stress Specification Method**.

Acoustic Wave Model

contains settings for treatment of acoustic pressure waves at the boundary.

Off

disables special treatment of acoustic pressure waves at the boundary.

Non Reflecting

enables the general non-reflecting boundary condition treatment described in General Non-Reflecting Boundary Conditions (p. 416).

Impedance

enables the impedance boundary condition treatment described in Impedance Boundary Conditions (p. 422).

Transparent Flow Forcing

enables the transparent flow forcing boundary condition described in Transparent Flow Forcing Boundary Conditions (p. 425).

Impedance Parameters

contains the parameters for the impedance boundary condition treatment. For details refer to Using the Impedance Boundary Condition (p. 424).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Transparent Flow Forcing Parameters
contains the parameters for the transparent flow forcing boundary condition treatment. For details refer to Using the Transparent Flow Forcing Boundary Condition (p. 426).

Thermal
contains the thermal parameters.

Temperature
specifies the static temperature of the flow.

Radiation
contains the boundary conditions for the radiation model at the velocity inlet.

External Black Body Temperature Method, Internal Emissivity
set the radiation boundary conditions when you are using the P-1, DTRM, DO, S2S, or MC models for radiation heat transfer. See Defining Boundary Conditions for Radiation (p. 869) for details.

Participates in Solar Ray Tracing
specifies whether or not the velocity inlet participates in solar ray tracing.

Solar Transmissivity Factor
specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the velocity inlet.

Participates in View Factor Calculation
specifies whether or not the velocity inlet participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

Species
contains the species parameters.

Specify Species in Mole Fractions
allows you to specify the species in mole fractions rather than mass fractions.

Species Mass Fractions
contains inputs for the mass fractions of defined species. See Defining Cell Zone and Boundary Conditions for Species (p. 990) for details about these inputs. These items will appear only if you are modeling non-reacting multi-species flow or you are using the finite-rate reaction formulation.

Mean Mixture Fraction, Mixture Fraction Variance
set inlet values for the PDF mixture fraction and its variance. These items will appear only if you are using the non-premixed or partially premixed combustion model.

Secondary Mean Mixture Fraction, Secondary Mixture Fraction Variance
set inlet values for the secondary mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model with two mixture fractions.)

Progress Variable
sets the value of the progress variable for premixed turbulent combustion. See Setting Boundary Conditions for the Progress Variable (p. 1094) for details.

This item will appear only if the premixed or partially premixed combustion model is used.

DPM
contains the discrete phase parameters.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discrete Phase BC Type
sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect
rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See Particle Reflection at Wall in the *Fluent Theory Guide*.)

trap
terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 23.25: "Trap" Boundary Condition for the Discrete Phase (p. 1303).

escape
reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 23.26: "Escape" Boundary Condition for the Discrete Phase (p. 1303).

wall-jet
indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 16.6: "Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

user-defined
specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function
sets the user-defined function from the drop-down list.

Multiphase
contains the multiphase parameters.

Volume Fraction
specifies the volume fraction of the secondary phase selected in the Boundary Conditions Task Page (p. 2389). This section of the dialog box will appear when one of the multiphase models is being used. See Defining Multiphase Cell Zone and Boundary Conditions (p. 1391) for details.

Secondary Phase for Inlet
specifies the secondary phase to which the parameters are applied. In case of a three-phase flow, select the corresponding secondary phase from this list.

Wave BC Options
allows you to choose between **Shallow Waves**, **Shallow/Intermediate Waves** or **Short Gravity Waves**. Information about the two types of waves is available in Open Channel Wave Boundary Conditions in the Theory Guide.

Free Surface Level
is determined by using the absolute value of height from the free surface to the origin in the direction of gravity and applying the correct sign based on whether the free surface level is above (positive) or below (negative) the origin.

Bottom Level
is valid for shallow and intermediate waves. The bottom level is used for calculating the liquid height.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reference Wave Direction
is the direction of wave propagation for a zero wave heading angle.

Wave Modeling Options
determines whether to use wave theory or a wave spectrum to model waves in the shallow/intermediate and short gravity wave regimes.

Number of Waves
specifies the number of waves you are defining when modeling waves using wave theory.

Wave Theory
allows you to choose from **First Order Airy** (the default), **Second Order Stokes**, **Third Order Stokes**, **Fourth Order Stokes**, and **Fifth Order Stokes** for shallow/intermediate waves and short gravity waves. For shallow waves, you can choose between **Fifth Order Solitary** and **Fifth Order Cnoidal**. Information about the types of wave theory is available in Open Channel Wave Boundary Conditions in the Theory Guide.

Wave Height
is the height difference between a wave crest to the neighboring trough.

Since a solitary wave does not have troughs, the wave height is the distance between a wave crest to mean free surface level.

Wave Length
is the wave length of the shallow wave or short gravity wave.

Inlet Offset Distance
is the translational distance from the reference point origin in the reference wave propagation direction. This option is used to generate a wave from a location other than the reference frame origin.

Phase Difference
is the phase difference between one wave and another.

Wave Heading Angle
is the angle between the direction of the wavefront and the reference wave direction, in the plane of the flow surface. In 2D, there are only two possibilities, zero degrees when the wave is in the reference wave direction, and 180 degrees when the wave is in the opposite direction.

Frequency Spectrum Method
specifies the spectrum to use. You can select **Pierson-Moskowitz** (appropriate for fully-developed seas), **Jonswap** (appropriate for fetch-limited seas), or **TMA** (appropriate for fetch-limited, finite-depth seas).

Peak Shape Parameter
controls the shape and amplitude of the frequency peak in the **Jonswap** and **TMA** formulations. This corresponds to γ in Equation 17.100 in the *Fluent Theory Guide*.

Significant Wave Height
is the mean wave height of the largest 1/3 of waves. This corresponds to H_s in Equation 17.99 in the *Fluent Theory Guide*.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Peak Wave Frequency
is the wave frequency corresponding to the highest wave energy. This corresponds to ω_p in Equation 17.99 and Equation 17.100 in the *Fluent Theory Guide*. ω_p can be expressed in terms of the peak wave period, T_p, as $\omega_p = {}^{2\pi}/_{T_p}$.

Minimum/Maximum Wave Frequency
specify the frequency range for the spectrum. Note that the **Peak Wave Frequency** must fall in between **Minimum Wave Frequency** and **Maximum Wave Frequency**.

Number of Frequency Components
specifies the number of components into which the frequency spectrum is divided.

Direction Spreading Method
specifies the method to use for specifying the directional spreading characteristics. You can choose **Unidirectional** (for long-crested waves), **Frequency Independent Cosine Function** (for short-crested waves where the directional component does not depend on frequency), and **Frequency Dependant Hyperbolic Function** (for short-crested waves where the directional component depends on frequency).

Frequency Independent Cosine Exponent
specifies the exponent in the Frequency Independent Cosine Function formulation. This corresponds to s in Equation 17.104 in the *Fluent Theory Guide*.

Mean Wave Heading Angle
specifies the principal wave heading direction.

Angular Spread
specifies the deviation from the mean wave direction for calculating the angular range.

Number of Angular Components
specifies the number of components into which the angular range is divided.

Potential
displays the boundary conditions for the electric potential field. This tab is available only if you have enabled either the **Electric Potential model** or the **Electrochemical** reaction model in the Species Dialog Box (p. 2332).

Potential Boundary Condition
is a drop-down list of available potential boundary condition types: **Specified Flux** and **Specified Value**. For the **Specified Flux** boundary condition, you will need to specify **Current Density** at the wall. For the **Specified Value** boundary condition, you will need to specify **Potential** at the wall.

UDS
contains the UDS parameters.

User-Defined Scalar Boundary Condition
appears only if user-defined scalars are specified.

User Scalar n
specifies whether the scalar is a specified flux or a specified value.

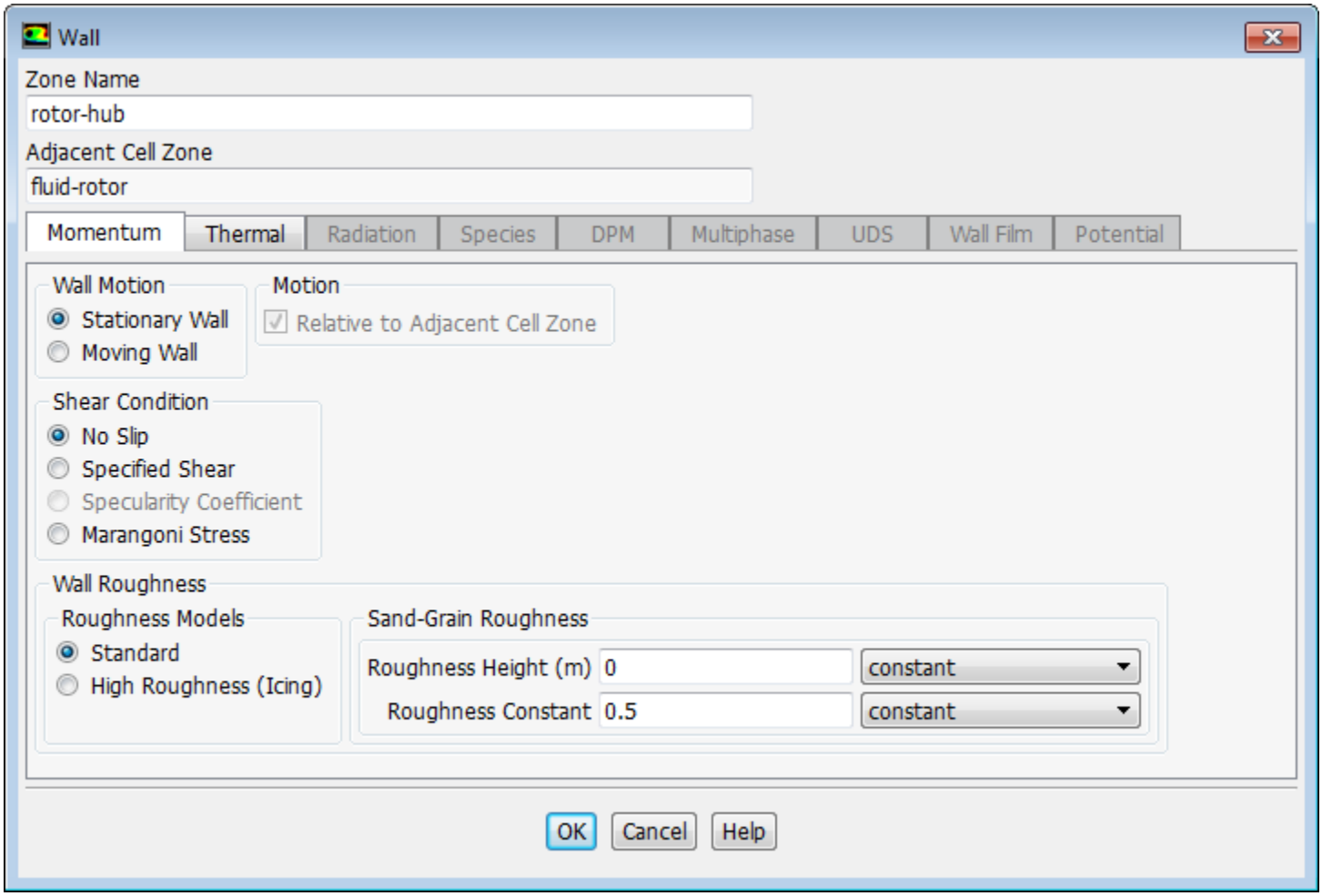
User-Defined Scalar Boundary Value
appears only if user-defined scalars are specified.

User Scalar n
specifies the value of the scalar.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.8.22. Wall Dialog Box

The **Wall** dialog box sets the boundary conditions for a wall zone. It is opened from the Boundary Conditions Task Page (p. 2389). See Inputs at Wall Boundaries (p. 358) for details about defining the items below.



Controls

Zone Name
sets the name of the zone.

Phase
displays the name of the phase. This item appears if the VOF, mixture, or Eulerian multiphase model is being used.

Adjacent Cell Zone
shows the name of the cell zone adjacent to the wall. (This is for informational use only; you cannot edit this field.)

Momentum
displays the momentum boundary conditions.

Wall Motion
contains options for specifying whether or not the wall is moving.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stationary Wall
specifies that the wall is not moving relative to the adjacent cell zone.

Moving Wall
enables specification of the tangential wall motion. Tangential wall motion is applicable only to viscous flows. Since the inviscid slip condition decouples the tangential wall velocity from the governing equations, tangential wall motion has no effect on inviscid flow.

Motion
contains inputs related to wall motion. See Velocity Conditions for Moving Walls (p. 359) for details.

Relative to Adjacent Cell Zone
enables the specification of a wall velocity relative to the velocity of the adjacent cell zone.(If the adjacent cell zone is not moving, this is equivalent to **Absolute**.)

Absolute
enables the specification of an absolute wall velocity,

Translational
enables the specification of a translational wall velocity.

Rotational
enables the specification of a rotational wall velocity.

Components
enables the specification of wall velocity components.

Speed
sets the translational or rotational speed of the wall (depending on whether you selected **Translational** or **Rotational**).

Direction
sets the direction vector of the translational velocity. (This item will appear if you have chosen the **Translational** option.)

Rotation-Axis Origin
sets the coordinates of the origin of the axis of rotation, thereby determining the location of the axis. (This item will appear if you have chosen the **Rotational** option for a non-axisymmetric case.)

Rotation-Axis Direction
sets the direction vector for the axis of rotation. (This item will appear if you have chosen the **Rotational** option for a non-axisymmetric case.)

Velocity Components
sets the **X**, **Y**, and **Z-Velocity** components of the wall motion. (This item will appear if you have chosen the **Components** option.)

Shear Condition
contains options for specifying the shear conditions at the wall.

No Slip
specifies a no-slip condition at the wall. No further inputs are required.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specified Shear
enables specification of zero or nonzero shear. See Specified Shear (p. 362) for details. This option is not available for moving walls.

Marangoni Stress
enables the specification of shear stress caused by the variation of surface tension due to temperature. This option is not available for moving walls.

Shear Stress
contains inputs related to wall shear. These items will appear when **Specified Shear** is selected as the **Shear Condition**. See Specified Shear (p. 362) for details.

X-Component, Y-Component, Z-Component, Swirl Component
specify the x, y, and z or swirl components of shear for a slip wall. **Swirl Component** is available only for axisymmetric swirl cases.

Specularity Coefficient
is used in multiphase granular flow. You can specify the specularity coefficient such that when the value is zero, this condition is equivalent to zero shear at the wall, but when the value is near unity, there is a significant amount of lateral momentum transfer.

Specularity Coefficient
allows you to enter a value between zero and one, which controls the amount of lateral momentum transfer.

Marangoni Stress
contains inputs related to Marangoni stress. This item will appear when **Marangoni Stress** is selected as the **Shear Condition**. See Marangoni Stress (p. 363) for details.

Surface Tension Gradient
specifies the surface tension gradient with respect to temperature ($d\sigma / dT$ in Equation 6.103 (p. 363)).

Wall Roughness
contains various models for defining wall roughness size/height in turbulent calculations.

Standard
uses the standard wall roughness methods for most turbulence models. See Wall Roughness Effects in Turbulent Wall-Bounded Flows (p. 364) for details.

High Roughness (Icing)
contains various models for use only with the Spalart-Allmaras and SST k-ω turbulence models. Specifically applicable to icing simulations, choose from **Specified Roughness**, **NASA Correlation**, **Shin-et-al**, or **ICE3D Roughness File** methods to specify the **Sand-Grain Roughness**. See Additional Roughness Models for Icing Simulations (p. 368) for details.

Wall Adhesion
contains inputs related to wall adhesion. This section of the dialog box will appear if you are using the VOF model and have enabled wall adhesion in the e Phase Interaction Dialog Box (p. 2363).

Contact Angles
specifies the contact angle at the wall for each pair of phases (θ_w in Figure 24.31: Measuring the Contact Angle (p. 1442) in the Theory Guide). See Steps for Setting Boundary Conditions (p. 1391) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Thermal

contains the thermal parameters. This tab is available only when the energy equation is turned on.

Thermal Conditions

contains radio buttons for selecting the thermal boundary condition type. See Thermal Boundary Conditions at Walls (p. 370) for details about these inputs:

Heat Flux

selects a specified heat flux condition.

Temperature

selects a specified wall temperature condition.

Convection

selects a convective heat transfer boundary condition model.

Radiation

selects an external radiation boundary condition.

Mixed

selects a combined convection/external radiation boundary condition.

Coupled

selects a coupled heat transfer condition. It is applicable only to walls that form the interface between two regions (such as the fluid/solid interface for a conjugate heat transfer problem).

via System Coupling

selects a heat transfer condition where a boundary can receive thermal data (either wall temperature or heat flow) from the System Coupling service. It is applicable when ANSYS Fluent is coupled with another system in Workbench using System Coupling (see Heat Transfer Boundary Conditions Through System Coupling (p. 377) for details).

via Mapped Interface

specifies that the thermal data is interpolated across a mapped interface. It is available only for the interface wall boundary zones created as part of a mapped interface (see Heat Transfer Boundary Conditions Across a Mapped Interface (p. 378) for details).

Once a condition type has been selected, the appropriate conditions can be specified.

Heat Flux

sets the wall heat flux to be used for the **Heat Flux** condition. A specification of zero **Heat Flux** is simply the adiabatic condition (no heat transfer). A positive value of heat flux implies that heat enters *into* the domain.

Temperature

sets the wall temperature to be used for the **Temperature** condition.

Heat Transfer Coefficient

sets the convective heat transfer coefficient to be used for the **Convection** condition (h_{eff} in Equation 6.122 (p. 383)).

Free Stream Temperature

sets the reference or free stream temperature to be used for the **Convection** condition (T_{ext} in Equation 6.122 (p. 383)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

External Emissivity
sets the emissivity of the external wall to be used for the **Radiation** condition (ε_{ext} in Equation 6.123 (p. 383)).

External Radiation Temperature
sets the temperature of the external radiation source/sink to be used for the **Radiation** condition (T_∞ in Equation 6.123 (p. 383)).

Internal Emissivity
sets the internal emissivity of the wall. This item will appear only if you are using the gray P-1, DO, or MC models, the DTRM, or the S2S model for radiation heat transfer. (Note that it will not appear if you are using the non-gray P-1, DO, or MC model. In these cases, you will enter the **Internal Emissivity** for each band in the **Radiation** tab.)

Wall Thickness
sets the thickness of a thin wall for the calculation of thermal resistance. (See Thin-Wall Thermal Resistance Parameters (p. 373) for details.)

Heat Generation Rate
sets the rate of heat generation in a wall without shell conduction.

Contact Resistance
sets the contact resistance (R_c in Equation 18.23 in the Theory Guide) at the wall. See Modeling Solidification and Melting (p. 1527) for details. This item appears only when the solidification/melting model is used.

Material Name
sets the material type for a thin wall. The conductivity of the material is used for the calculation of thin-wall thermal resistance. (See Thin-Wall Thermal Resistance Parameters (p. 373) for details.) **Material** is used only when **Wall Thickness** is nonzero. Materials are defined with the Materials Task Page (p. 2300).

Shell Conduction
enables shell conduction for the wall. See Shell Conduction (p. 376) for details.

Edit...
is only available when **Shell Conduction** is enabled, and opens the Shell Conduction Layers Dialog Box (p. 2804) in order to allow you to define the shell conduction settings for the wall.

Radiation
displays the boundary conditions for the S2S model, the DO model, the MC model, and the non-gray P-1 model at the wall. This tab is only available if you are using the surface to surface model, the discrete ordinates model, the monte carlo model, or the non-gray P-1 model. See Forming Surface Clusters (p. 853), Wall Boundary Conditions for the DO Model (p. 871), and Wall Boundary Conditions for the DTRM, and the P-1, S2S, and Rosseland Models (p. 870) for details.

BC Type
contains a drop-down list of available radiation boundary condition types. The available options are **opaque** and **semi-transparent**. This item will appear only if you are using the discrete ordinates model.

Internal Emissivity
specifies the internal emissivity of the wall in each wavelength band. This item will appear only if you are using the non-gray discrete ordinates model and you have selected **opaque** as the **BC Type**, or if you are using the non-gray P-1 model.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Diffuse Fraction

specifies the fraction of the irradiation that is to be treated as diffuse. By default, the **Diffuse Fraction** is set to 1, indicating that all of the irradiation is diffuse. If the non-gray DO model is being used, the **Diffuse Fraction** can be specified for each band. This item will appear only if you are using the Monte Carlo or discrete ordinates model.

Beam Width

specifies the beam width for an external semi-transparent wall in terms of the **Theta** and **Phi** extents. This item will appear only if you are using the discrete ordinates radiation model and you have selected **semi-transparent** as the **BC Type**.

Beam Direction

specifies the beam direction as an **X,Y,Z** vector. You can specify the **Beam Direction** as a constant, a profile, or a user-defined function. This item will appear only if you are using the Monte Carlo model or the discrete ordinates radiation model and you have selected **semi-transparent** as the **BC Type**.

Direct Irradiation

specifies the value of the irradiation flux. If the non-gray DO model is being used, a constant **Direct Irradiation** can be specified for each band.

This item will appear only if you are using the Monte Carlo model or the discrete ordinates radiation model and you have selected **semi-transparent** as the **BC Type**.

Apply Direct Irradiation Parallel to the Beam

is the default means of specifying the scale of irradiation flux. When enabled, ANSYS Fluent assumes that the value of **Direct Irradiation** that you specify is the irradiation flux parallel to the **Beam Direction**. When deselected, ANSYS Fluent instead assumes that the value specified is the flux parallel to the face normals and will calculate the resulting beam parallel flux for every face. This item will appear only if you are using the discrete ordinates model.

Diffuse Irradiation

specifies the value of the irradiation flux. If the non-gray DO model is being used, a constant **Diffuse Irradiation** can be specified for each band.

This item will appear only if you are using the discrete ordinates radiation model and you have selected **semi-transparent** as the **BC Type**.

Solar Boundary Conditions

contains the settings for solar ray tracing. This group box is available only if you select **Solar Ray Tracing** from the **Model** list in the **Solar Load** group box of the **Radiation Model** dialog box. For semitransparent coupled walls, the solar ray tracing settings are defined on the original wall and not on the shadow wall. See Solar Ray Tracing (p. 900) for details.

Participates in Solar Ray Tracing

specifies whether or not the wall participates in solar ray tracing.

Absorptivity

contains the settings that define the absorptivity of wall.

Direct Visible

specifies a multiplier (ranging from 0 to 1) that is applied to the visible portion of the direct solar radiation spectrum to account for the absorption of the wall. The value should be defined for normal incident rays.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Direct IR
specifies a multiplier (ranging from 0 to 1) that is applied to the infrared portion of the direct solar radiation spectrum to account for the absorption of the wall. The value should be defined for normal incident rays.

Diffuse Hemispherical
specifies a multiplier (ranging from 0 to 1) that is applied to the diffuse solar radiation to account for the absorption of the wall. This setting is only available for semi-transparent walls.

Transmissivity
contains the settings that define the transmissivity of wall. This group box is only available when **semi-transparent** is selected for **BC Type**.

Direct Visible
specifies a multiplier (ranging from 0 to 1) that is applied to the visible portion of the direct solar radiation spectrum to account for the transmissivity of the wall. The value should be defined for normal incident rays. This setting is only available for semi-transparent walls.

Direct IR
specifies a multiplier (ranging from 0 to 1) that is applied to the infrared portion of the direct solar radiation spectrum to account for the transmissivity of the wall. The value should be defined for normal incident rays. This setting is only available for semi-transparent walls.

Diffuse Hemispherical
specifies a multiplier (ranging from 0 to 1) that is applied to the diffuse solar irradiation to account for the transmissivity of the wall. This setting is only available for semi-transparent walls.

S2S Parameters
contains the settings for the S2S radiation model. This group box is available only if you select the **Surface to Surface** radiation model. See Forming Surface Clusters (p. 853) for details.

Faces Per Surface Cluster
sets the number of faces per surface cluster (FPSC) for the wall, and thus controls the number of radiating surfaces and (if you select **Cluster to Cluster** for **Basis** in the **View Factors and Clustering** dialog box) view factor surfaces.

Critical Zone
specifies that the wall is a critical zone. When this option is enabled, the value entered for **Faces Per Surface Cluster** will not be altered when you use **Automatic** clustering in the **View Factors and Clustering** dialog box, and impacts the calculations and actions performed by the buttons in the **Maximum Distance from Critical Zone** group box of the **Participating Boundary Zones** dialog box.

Participates in View Factor Calculation
specifies whether or not the wall participates in the view factor calculation as part of the S2S radiation model. This option is available only if you select the **Surface to Surface** radiation model.

Species
contains the species parameters. This tab is available only if you have enabled the **Species Transport** model in the Species Model Dialog Box (p. 2216).

Reaction
activates reactions at the wall. This item will appear only if you have enabled any of the reactions in the Species Model Dialog Box (p. 2216).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reaction Mechanisms
allows you to specify a defined group, or mechanism, of available reactions. This item will appear only if the **Reaction** option has been turned on. See Defining Zone-Based Reaction Mechanisms (p. 984) for details about defining reaction mechanisms.

Surface Area Washcoat Factor
allows you to specify a factor, which multiplies the wall area to account for the increased surface area of washcoats. This item will appear only if the **Reaction** option has been turned on. See Species Boundary Conditions for Walls (p. 379) for details.

Species Boundary Condition
contains options for the specification of species boundary conditions. See Species Boundary Conditions for Walls (p. 379) for details.

Zero Diffusive Flux
indicates a zero-flux condition for a species. This is the default condition.

Specified Mass Fraction
indicates that the species mass fraction will be specified.

Species Mass Fractions
contains inputs for the species mass fractions of any species for which you have selected **Species Mass Fraction** as the **Species Boundary Condition**.

DPM
contains the discrete phase parameters. This tab is available only if you have defined at least one injection.

Discrete Phase Model Conditions
contains inputs for setting the fate of particle trajectories at the wall. These options will appear when one or more injections have been defined. See Setting Boundary Conditions for the Discrete Phase (p. 1301) for details.

Boundary Cond. Type
sets the way that the discrete phase behaves with respect to the boundary.

reflect
rebounds the particle off the boundary with a change in its momentum as defined by the coefficients of restitution (see Particle Reflection at Wall in the *Fluent Theory Guide*). When at least one injection accounts for the rough wall model (the **Rough Wall Model** option is selected under the **Physical Models** tab in the **Set Injections Properties** dialog box), you can specify the **DPM Wall Roughness Parameters** (**Ra**, **Rq**, and **RSm** in Equation 16.216 in the *Fluent Theory Guide*). (See Rough Wall Model (p. 1292)).

trap
terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 23.25: "Trap" Boundary Condition for the Discrete Phase (p. 1303).

escape
reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 23.26: "Escape" Boundary Condition for the Discrete Phase (p. 1303).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

wall-jet
indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 16.6:"Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

wall-film
allows you to enable **Film Condensation** (Film Condensation Model (p. 1307)), **Wall Boundary Layer Model** (Wall Boundary Layer Model (p. 1309)), and specify the **Film Model Parameters**:

Number Of Splashed Drops
is used to model splashing. The minimum number of splashed parcels is three. A value of zero implies no splashing.

Critical Temperature Factor
is used to determine the transition temperature (Equation 16.222 and Equation 16.250 in the *Fluent Theory Guide* for the **stanton-rutland** and **kuhnke** splashing models, respectively).

Impingement/Splashing Model

stanton-rutland
(default) consists of four regimes, namely, stick, rebound, spread, and splash, which are based on the impact energy and wall temperature. See The Stanton-Rutland Model in the *Fluent Theory Guide* for more details.

kuhnke
consists of four regimes: spread, rebound, splash, and dry splash (thermal breakup). See The Kuhnke Model in the *Fluent Theory Guide* for more details. You must specify the following **DPM Wall Roughness Parameters**:

Roughness Ra
is used to compute K_{cold} in Equation 16.258 in the *Fluent Theory Guide*.

Roughness Rz
Roughness Rz: is used to compute splashed droplets ejection angle in Equation 16.270 in the *Fluent Theory Guide*.

user-defined
specifies a user-defined function to define the discrete phase boundary condition type.

Boundary Cond. Function
sets the user-defined function from the drop-down list.

Discrete Phase Reflection Coefficients
determine the behavior of reflecting particles. This item appears when **reflect** is chosen as the **Boundary Cond. Type**. See Discrete Phase Boundary Condition Types (p. 1301) for details on setting the following items.

Normal
sets the type of function for the normal coefficient of restitution. This function can be **constant**, **piecewise-linear**, **piecewise-polynomial**, or **polynomial**.

Tangent
sets the type of function for the tangential coefficient of restitution. This function can be **constant**, **piecewise-linear**, **piecewise-polynomial**, or **polynomial**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DEM Collision Partner
contains a list of names to designate the collision partner.

Erosion Model
contains a list of erosion models. See Setting Particle Erosion and Accretion Parameters (p. 1309) for details about these items.

Generic Model
specifies erosion rates to be computed using the generic erosion model. (See Accretion in the *Fluent Theory Guide* for background information about this model.) When this model is enabled, you must specify the erosion parameters in the **Generic Erosion Model Parameters** dialog box that opens by clicking the corresponding **Edit...** button as described in Setting Particle Erosion and Accretion Parameters (p. 1309).

Finnie
specifies erosion rates to be computed using the Finnie formulation. (See Accretion in the *Fluent Theory Guide* for background information about this model.) When this model is enabled, you must specify the erosion parameters in the **Finnie Model Parameters** dialog box that opens by clicking the corresponding **Edit...** button as described in Setting Particle Erosion and Accretion Parameters (p. 1309).

McLaury
specifies erosion rates to be computed using the McLaury formulation. (See McLaury Erosion Model in the *Fluent Theory Guide* for background information about this model.) When this model is enabled, you must specify the erosion parameters in the **McLaury Model Parameters** dialog box that opens by clicking the corresponding **Edit...** button as described in Setting Particle Erosion and Accretion Parameters (p. 1309).

Oka
specifies erosion rates to be computed using the Oka formulation. (See Oka Erosion Model in the *Fluent Theory Guide* for background information about this model.) When this model is enabled, you must specify the erosion parameters in the **Oka Model Parameters** dialog box that opens by clicking the corresponding **Edit...** button as described in Setting Particle Erosion and Accretion Parameters (p. 1309).

Friction Coefficient
specifies μ in Equation 16.209 and Equation 16.214 in the *Fluent Theory Guide* as a **constant** or a **piecewise-linear**, **piecewise-polynomial**, or **polynomial** function of relative velocity. This item appears only when **Enable Rotation** is selected under the **Physical Model** tab of the Set Injection Properties Dialog Box (p. 2796).

UDS
displays the boundary conditions for user-defined scalars (UDSs) at the wall. This tab is available only if you have specified a nonzero number of user-defined scalars in the User-Defined Scalars Dialog Box (p. 2829).

User Defined Scalar Boundary Condition
contains options for the specification of UDS boundary conditions. See the separate Fluent Customization Manual for details.

Specified Flux
indicates that the flux of the UDS at the wall will be specified.

Specified Value
indicates that the value for the UDS at the wall will be specified.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

User Defined Scalar Boundary Value
contains inputs for the value of the flux of the UDS, or the value of the UDS itself, depending on your selection for that UDS under **User Defined Scalar Boundary Condition**.

Wall Film
displays the boundary conditions for liquid films at the wall. This tab is available only if you have enabled the Eulerian Wall Film model in the Models Task Page (p. 2169).

Eulerian Film Wall
allows you to define a film wall condition for any wall. See Modeling Eulerian Wall Films (p. 1535) for details. Once this option is enabled, you can set the following:

Film Condition Type
allows you to select the desired condition type.

Boundary Condition
allows you to specify settings for boundary **Film Conditions**:

Film Mass Flux
The film mass source in terms of mass flux per unit area ($kg/m^2–s$).

X-Momentum Flux, Y-Momentum Flux, and Z-Momentum Flux
The film momentum source in terms of momentum flux per unit area (N/m^2).

Incoming Film Temperature
The film temperature (K).

Film Passive Scalar Flux
The film passive scalar source in terms of mass flux per unit area ($kg/m^2–s$).

Initial Condition
allows you to specify settings for initial **Film Conditions**:

Film Height
The film height at the wall boundary.

X-Velocity, Y-Velocity, and Z-Velocity
The film velocity components at the wall boundary.

Film Temperature
The film temperature (K).

Film Passive Scalar
The film passive scalar at the wall boundary (dimensionless).

Relative Film Momentum Flux
(moving reference frames or moving walls only) enables specification of the film momentum flux relative to the wall on which liquid film is defined (**Boundary Condition** type only). This option is only available when **Moving Wall** is selected in the **Wall** dialog box and/or **Frame Motion** is selected in the **Fluid** dialog box.

Relative Initial Film Velocity
(moving reference frames or moving walls only) enables specification of the initial velocity relative to the wall on which liquid film is defined (**Initial Condition** type only). This option is only available

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

when **Moving Wall** is selected in the **Wall** dialog box and/or **Frame Motion** is selected in the **Fluid** dialog box.

User Source Terms

enables you to specify the additional source terms to the film continuity (**Mass Flux**), momentum (**X-Momentum Flux**, **Y-Momentum Flux**, and **Z-Momentum Flux**), energy (**Heat Flux**), and passive scalar equations (**Scalar Flux**). This option is only available with initial conditions.

Flow Momentum Coupling

When this option is selected, the liquid film and the gas flow will share the same velocity at the interface of the liquid-gas interface using a two-way coupling. When this option is not selected, the coupling between the liquid film and the gas flow is only one-way, namely, while the gas flow impacts the film flow, the film flow does not impact the bulk of the gas flow.

Film Phase Change

enables the phase change for the selected wall. This option is available only when **Phase Change** is selected in the **Model Options and Setup** tab of the Eulerian Wall Film Dialog Box (p. 2294). Once enabled, you can specify the following **Phase Change Parameters** (see Coupling of Wall Film with Mixture Species Transport in the Theory Guide):

Condensation Rate Constant

is the value of C_{con} used in Equation 19.28 in the Theory Guide. You can also specify the condensation rate via a user-defined profile UDF by selecting the name of the compiled UDF from the drop-down list.

Vaporization Rate Constant

is the value of C_{vap} used in Equation 19.28 in the Theory Guide. You can also specify the vaporization rate via a user-defined profile UDF by selecting the name of the compiled UDF from the drop-down list.

Impingement Parameters

contains parameters related to particle-wall impingement

Impingement Model

is a drop-down list that contains available impingement models:

- **stanton-rutland** (default): consists of four impingement regimes: stick, rebound, spread and splash, which are based on the impact energy and wall temperature. See The Stanton-Rutland Model in the *Fluent Theory Guide* for details.
- **kuhnke**: consists of four impingement regimes: stick, rebound, spread and splash (thermal breakup). See The Kuhnke Model in the *Fluent Theory Guide* for details.

DPM Wall Splash

allows you to enable the wall splashing for the selected wall and specify the **Number of Splashed Particles**. The minimum value is 3. This option is available when **Particle Splashing** is enabled in the **Eulerian Wall Film** dialog box.

Kuhnke Model Parameters

(available when the **kuhnke** model is selected from the **Impingement Model** drop-down list)

Wall Roughness Length

enables you to specify wall roughness values used by the Kuhnke model:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Ra**: the wall mean roughness that is used to compute K_{cold} in Equation 16.258 in the *Fluent Theory Guide*
- **Rz**: the average surface roughness that is used to compute splashed particle reflection angle distribution Equation 16.270 in the *Fluent Theory Guide*.

Critical Temperature Factor
is the dimensionless variable that is used to determine the impingement regime transition temperature in Equation 16.250 in the *Fluent Theory Guide*.

Potential
displays the boundary conditions for the electric potential field at the wall. This tab is available only if you have enabled either the **Electric Potential model** under the **Model** tree branch or the **Electrochemical** Reaction model in the Species Dialog Box (p. 2332).

Electrochemical Reaction
enables electrochemical reactions at the wall.

Faradaic Heat
(if enabled) includes heat from the electrochemical reaction in the energy equation.

Potential Boundary Condition
contains a drop-down list of available potential boundary condition types for the external wall: **Specified Flux** and **Specified Value**. For the **Specified Flux** boundary condition, you will need to specify **Current Density** at the wall. For the **Specified Value** boundary condition, you will need to specify **Potential** at the wall.

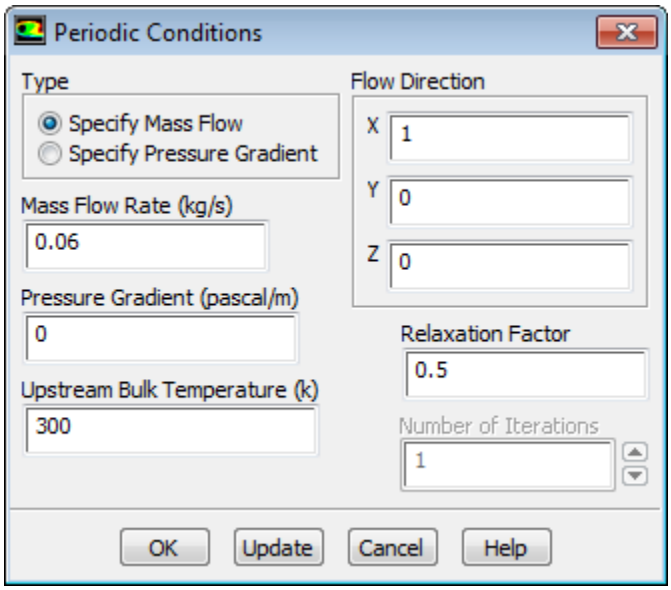
Reaction Mechanism
contains a drop-down list of all available electrochemical reaction mechanisms.

37.8.23. Periodic Conditions Dialog Box

The **Periodic Conditions** dialog box allows you to set parameters that define fully-developed periodic flow and heat transfer. See User Inputs for the Pressure-Based Solver (p. 584) and Using Periodic Heat Transfer (p. 914) for details.

(This dialog box is available only when the pressure-based solver is used; it is not available for the density-based coupled solvers.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Specify Mass Flow
enables the specification of the mass flow rate.

Specify Pressure Gradient
enables the specification of the pressure gradient.

Mass Flow Rate
specifies the mass flow rate. This item will not be available if you selected the **Specify Pressure Gradient** option.

Important

For axisymmetric problems, the mass flow rate is per 2π radians.

Pressure Gradient
specifies the pressure gradient (β in Equation 1.22 in the Theory Guide).

Upstream Bulk Temperature
sets the inlet bulk temperature for periodic heat transfer calculations.

Flow Direction
sets the direction of the periodic flow. The direction vector must be parallel to the periodic translation direction or its opposite.

Relaxation Factor
sets the under-relaxation factor that controls convergence of the iteration process described in Setting Parameters for the Calculation of β (p. 585) for specified mass flow.

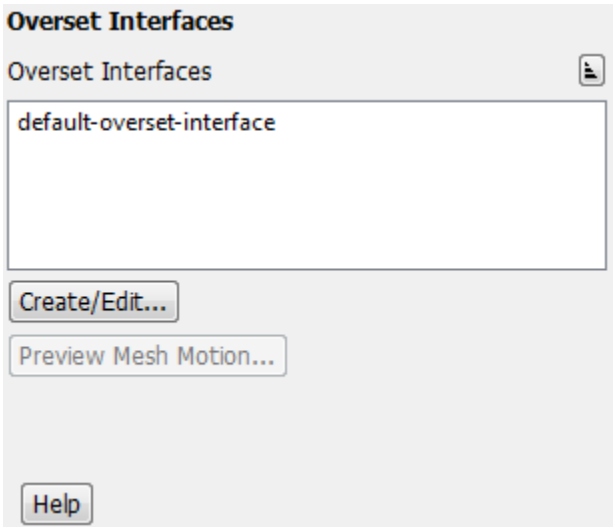
Number of Iterations
sets the number of subiterations done on the correction of β in the pressure correction equation for specified mass flow. See Setting Parameters for the Calculation of β (p. 585) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Update
updates the **Pressure Gradient** field with the current value.

37.9. Overset Interfaces Task Page

The **Overset Interfaces** task page allows you to create overset interfaces. For additional information about overset interfaces, see Overset Meshes (p. 172).



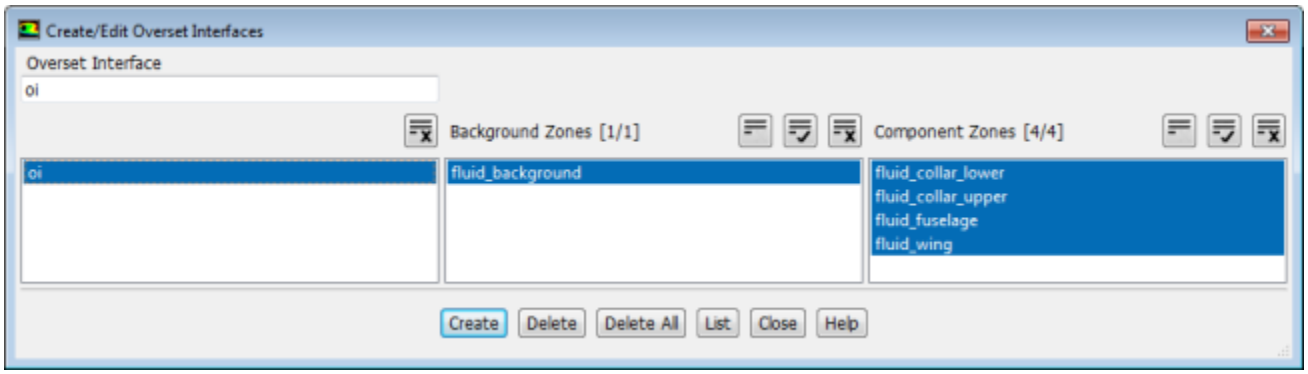
Controls

Create/Edit...
displays the Create/Edit Overset Interfaces Dialog Box (p. 2470).

Preview Mesh Motion...
displays the Mesh Motion Dialog Box (p. 2500).

37.9.1. Create/Edit Overset Interfaces Dialog Box

The **Create/Edit Overset Interfaces** dialog box allows you to create and delete overset interfaces. For additional information, see Overset Meshes (p. 172).



Overset Interface
contains a text entry box where you must specify the name of the overset interface. Below is a list of all existing overset interfaces.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Background Zones
lists all of the background zones available for selection when creating an overset interface. Multiple background zones can be part of an overset interface, however a background zone cannot be part of multiple overset interfaces.

Component Zones
lists all of the component zones available for selection when creating of an overset interface. Multiple components can be part of an overset interface, however a component zone cannot be part of multiple overset interfaces.

Create
creates an overset interface using the name in the **Overset Interface** text box and the zones selected under **Background Zones** and **Component Zones**. Note that grid priorities, if required, are added to the interface after it is created, using the text command `define/overset-interfaces/grid-priorities`.

Delete
deletes the selected **Overset Interface**.

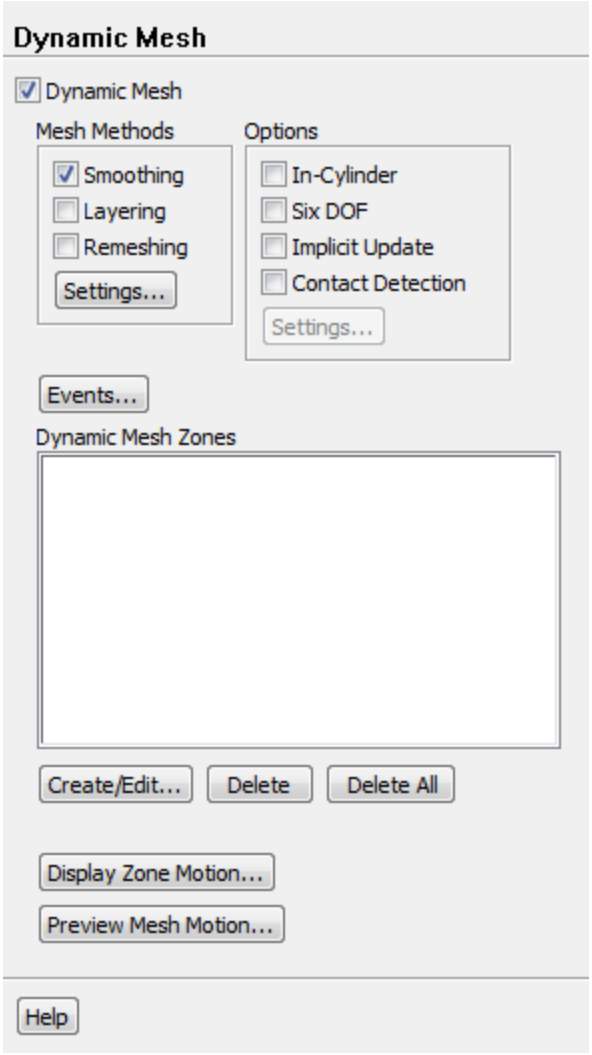
Delete All
deletes all overset interfaces in the domain.

List
prints zonal information and grid priorities for the selected **Overset Interface**.

37.10. Dynamic Mesh Task Page

The **Dynamic Mesh** task page allows you to define the all the parameters for modeling a dynamic mesh model. See Setting Dynamic Mesh Modeling Parameters (p. 643) for details about using the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Dynamic Mesh
enables the dynamic mesh model and activates the controls in the task page.

Mesh Methods
contains options to specify the mesh update method(s).

Smoothing
enables mesh smoothing. See Smoothing Methods (p. 644) for details about the smoothing methods available.

Layering
enables the dynamic layering method that can be used to add or remove layers of cells adjacent to a moving boundary based on the height of the layer adjacent to the moving surface in prismatic mesh zones. See Dynamic Layering (p. 661) for details.

Remeshing
enables the local or zonal remeshing methods. In local remeshing, the cells that violate the skewness or size criteria are agglomerated and locally remeshed; see Local Remeshing Method (p. 668) for further details. In zonal remeshing, the complete cell zone (including the boundary zones) is remeshed; see CutCell Zone Remeshing Method (p. 681) for further details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Settings...
displays the Mesh Method Settings Dialog Box (p. 2474) in which you can specify settings for the Smoothing, Layering, and Remeshing methods.

Options
contains options to specify specialized dynamic mesh models.

In-Cylinder
enables the in-cylinder model. See In-Cylinder Settings (p. 690) for more information.

Six DOF
enables the six degrees of freedom (DOF) solver. See Six DOF Solver Settings (p. 705) for more information.

Implicit Update
specifies that the mesh is updated during a time step (as opposed to just at the beginning of a time step). See Implicit Update Settings (p. 709) for more information.

Contact Detection
enables contact detection. See Contact Detection Settings (p. 711) for more information.

Settings...
opens the Options Dialog Box (p. 2478), where you can set the parameters for the options that are enabled in the **Options** group box.

Events...
displays the Dynamic Mesh Events Dialog Box (p. 2486).

Dynamic Mesh Zones
displays a list of dynamic mesh zones.

Create/Edit...
displays the Dynamic Mesh Zones Dialog Box (p. 2490).

Delete
removes the selected dynamic zone(s) from the **Dynamic Mesh Zones** list.

Delete All
removes all dynamic zones from the **Dynamic Mesh Zones** list.

Display Zone Motion...
displays the Zone Motion Dialog Box (p. 2499).

Preview Mesh Motion...
displays the Mesh Motion Dialog Box (p. 2500).

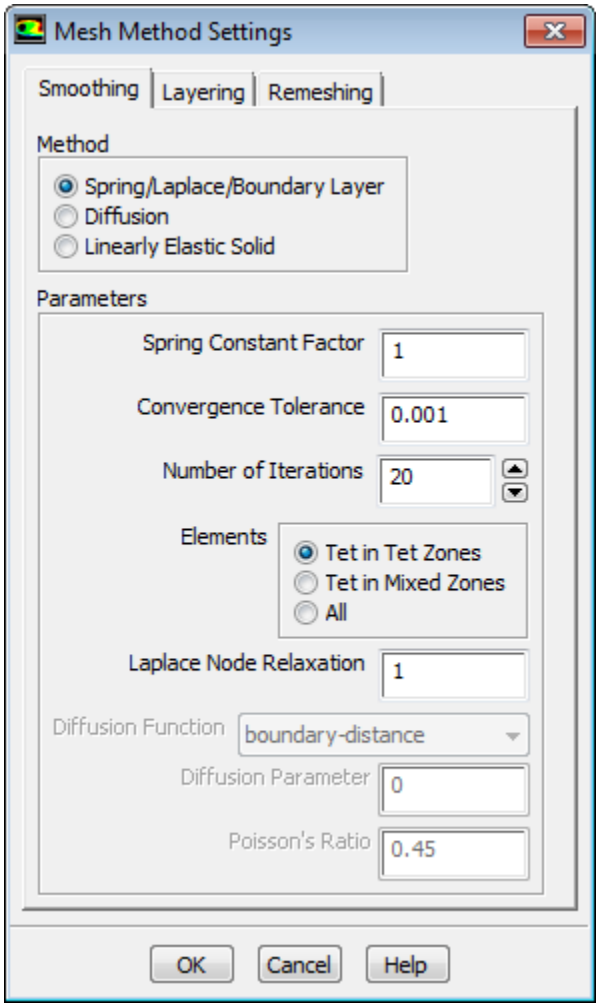
For additional information, see the following sections:
37.10.1. Mesh Method Settings Dialog Box
37.10.2. Mesh Scale Info Dialog Box
37.10.3. Options Dialog Box
37.10.4. In-Cylinder Output Controls Dialog Box
37.10.5. Six DOF Properties Dialog Box
37.10.6. Flow Controls Dialog Box
37.10.7. Dynamic Mesh Events Dialog Box
37.10.8. Define Event Dialog Box

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.10.9. Events Preview Dialog Box
37.10.10. Dynamic Mesh Zones Dialog Box
37.10.11. Orientation Calculator Dialog Box
37.10.12. Inflation Settings Dialog Box
37.10.13. CutCell Boundary Zones Info Dialog Box
37.10.14. Zone Scale Info Dialog Box
37.10.15. Zone Motion Dialog Box
37.10.16. Mesh Motion Dialog Box
37.10.17. Autosave Case During Mesh Motion Preview Dialog Box

37.10.1. Mesh Method Settings Dialog Box

The **Mesh Method Settings** dialog box allows you to apply settings for the smoothing, layering, or remeshing methods.



Smoothing

contains parameters to be specified for the smoothing mesh update method.

Method

allows you to specify the smoothing method.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Spring/Laplace/Boundary Layer
specifies that the smoothing method is spring based, or appropriate for the Laplacian smoothing method (for 2.5D remeshing) or the boundary layer smoothing method.

Diffusion
selects the diffusion-based smoothing method.

Linearly Elastic Solid
selects smoothing based on the equations for a linearly elastic solid.

Parameters
allows you to define the settings for the smoothing.

Spring Constant Factor
controls the spring stiffness.

Convergence Tolerance
controls the smoothing convergence.

Number of Iterations
specifies the number of iterations.

Elements
controls the elements where spring-based smoothing is applied.

Tet in Tet Zones
(3D only) only cell zones with all tetrahedral elements get smoothed.

Tet in Mixed Zones
(3D only) tetrahedral elements in mixed element zones get smoothed.

Tri in Tri Zones
(2D only) only cell zones with all triangular elements get smoothed.

Tri in MixedZones
(2D only) triangular elements in mixed element zones get smoothed.

All
all element types get smoothed.

Laplace Node Relaxation
specifies how the update of the node positions is relaxed on boundaries where there is Laplacian smoothing.

Diffusion Function
specifies whether the diffusion coefficient is a function of the **boundary-distance** (Equation 10.8 (p. 650)) or the **cell-volume** (Equation 10.9 (p. 650)). This drop-down list is only available when **Diffusion** is selected from the **Method** list.

Diffusion Parameter
specifies α in Equation 10.8 (p. 650) or Equation 10.9 (p. 650), depending on the selected **Diffusion Function**. This number-entry box is only available when **Diffusion** is selected from the **Method** list.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Poisson's Ratio
specifies the linearly elastic solid material property ν in Equation 10.12 (p. 656).

Layering
contains parameters to be specified for the layering mesh update method.

Options
specifies the criteria for splitting or collapsing cell layers.

Height Based
specifies that the cell layers are split or merged based on height.

Ratio Based
specifies that the cell layers are split or merged based on ratios.

Split Factor
specifies the value of α_s in Equation 10.15 (p. 662). It controls the height or ratio at which the cells are split.

Collapse Factor
specifies the value of α_c in Equation 10.16 (p. 663). It controls the height or ratio at which the cells are collapsed and merged into the next layer.

Remeshing
contains parameters to be specified for the remeshing mesh update method.

Remeshing Methods
contain options that control remeshing.

Local Cell
allows you to remesh deforming boundary cells.

Local Face
allows you to remesh deforming boundary faces. This option is available for 3D cases.

Region Face
allows you to remesh a region.

CutCell Zone
allows you to replace an entire cell zone with a predominantly Cartesian mesh. This option is only available for 3D cases. Note that the parameters that control this remeshing method are set in the **Dynamic Mesh Zones** dialog box. See Using the CutCell Zone Remeshing Method (p. 683) for more information.

2.5D
enables the 2.5D model. This option is only available for 3D cases. See Using the 2.5D Model (p. 686) for more information.

Parameters
contains parameters that control remeshing for all of the remeshing methods except for **CutCell Zone**.

Minimum Length Scale
specifies the lower limit of cell size below which the cells are marked for remeshing.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Maximum Length Scale
specifies the upper limit of cell size above which the cells are marked for remeshing.

Maximum Cell Skewness
specifies the desired maximum skewness for the mesh.

Maximum Face Skewness
specifies the desired maximum skewness for the surface mesh. This option is active, when **Local Face** is selected under **Remeshing Methods**.

Size Remeshing Interval
specifies the interval in time steps for remeshing based on the above size criteria only. Marking of cells based on skewness occurs automatically at every time step when **Remeshing** is enabled.

Mesh Scale Info...
opens the Mesh Scale Info Dialog Box (p. 2477), in which you can view the statistics of the mesh, such as the minimum and maximum length scale values and the maximum cell and face skewness values.

Use Defaults
resets the remeshing parameters to the default values.

Sizing Function
contains parameters that control the sizing function.

On
allows you to enable or disable the sizing function.

Resolution
sets the resolution for the sizing function. See Setting Dynamic Mesh Modeling Parameters (p. 643) for more information. This item will appear only if **Sizing Function** is enabled.

Variation
specifies the value of α in Equation 10.22 (p. 674). This item will appear only if **Sizing Function** is enabled.

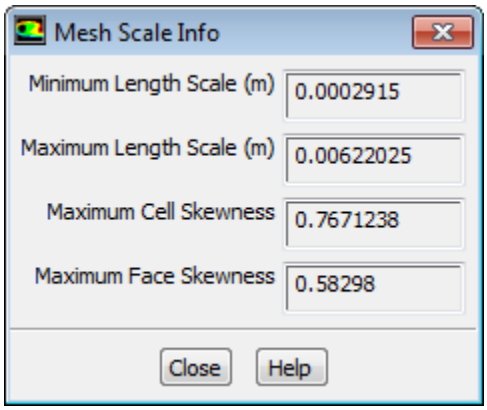
Rate
specifies the value of β in Equation 10.23 (p. 674). This item will appear only if **Sizing Function** is enabled.

Use Defaults
resets the sizing function parameters to the default values. This item will appear only if **Sizing Function** is enabled.

37.10.2. Mesh Scale Info Dialog Box

The **Mesh Scale Info** dialog box allows you to inspect the values of minimum and maximum length scale and maximum cell/face skewness in a mesh.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Minimum Length Scale
displays the lower limit of cell size below which the cells are marked for remeshing.

Maximum Length Scale
displays the upper limit of cell size below which the cells are marked for remeshing.

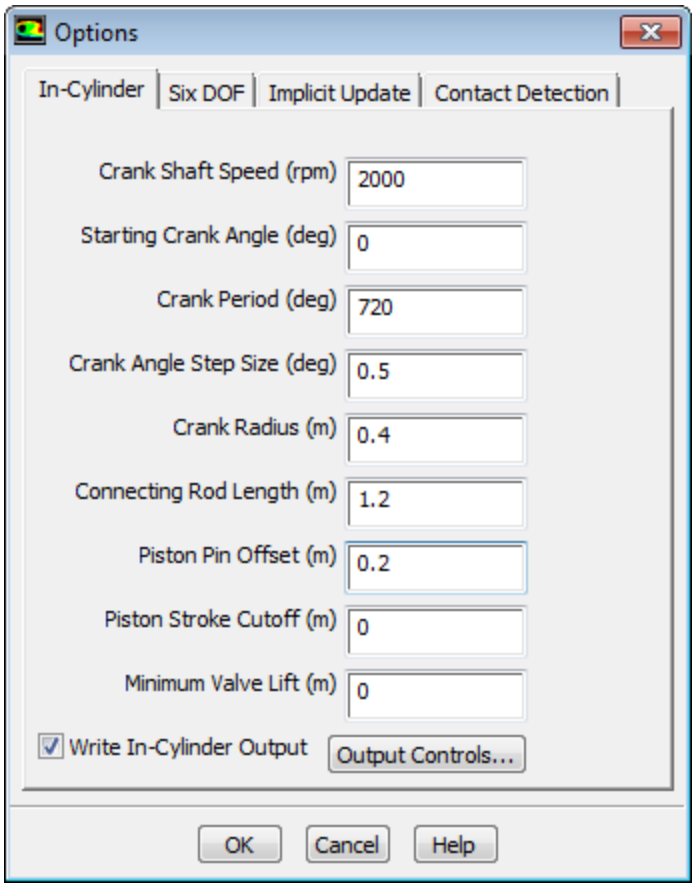
Maximum Cell Skewness
displays the maximum cell skewness in the zone.

Maximum Face Skewness
displays the maximum cell skewness in the surface mesh.

37.10.3. Options Dialog Box

The **Options** dialog box allows you to set the parameters for the options available in the **Options** group box of the **Dynamic Mesh** task page.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In-Cylinder

contains parameters to be specified for the in-cylinder model. See In-Cylinder Settings (p. 690) for more information.

Crank Shaft Speed

specifies the speed of the crank shaft.

Starting Crank Angle

specifies the starting crank angle.

Crank Period

specifies the crank period.

Crank Angle Step Size

specifies the crank angle step size used to determine the time step size to advance the solution.

Crank Radius

specifies the crank radius to calculate the piston location.

Piston Pin Offset

specifies the perpendicular offset of the piston pin from the plane defined by the crank shaft axis and the direction of motion of the piston. The sign of this value is positive if top-dead-center (TDC) occurs prior to a crank angle of 0°.

Connecting Rod Length

specifies the length of the connecting rod.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Piston Stroke Cutoff
specifies the piston stroke cutoff used to control the onset of layering in the cylinder chamber.

Minimum Valve Lift
specifies the minimum valve lift.

Write In-Cylinder Output
enables or disables the writing of in-cylinder specific output parameters. When this option is enabled, the **Output Controls...** button becomes active.

Output Controls...
displays the In-Cylinder Output Controls Dialog Box (p. 2481) and is available only after the **Write In-Cylinder Output** option is enabled.

Six DOF
contains parameters to be specified for the six degrees of freedom (DOF) solver. See Six DOF Solver Settings (p. 705) for more information.

Six DOF Properties
displays a list of defined sets of six DOF properties.

Create/Edit...
opens the Six DOF Properties Dialog Box (p. 2482).

Delete
removes the selected set of six DOF properties from the **Six DOF Properties** list.

Delete All
removes all sets of six DOF properties from the **Six DOF Properties** list.

Gravitational Acceleration
contains the text entry boxes for gravitational acceleration in X, Y, and Z directions.

X, Y, Z
specifies the gravitational acceleration in X, Y, and Z directions respectively.

Write Motion History
allows you to keep track of an object's motion history.

File Name
allows you to specify a file name for saving the object's motion history.

Implicit Update
contains parameters to be specified for implicit mesh updating. See Implicit Update Settings (p. 709) for more information.

Update Interval
allows you to specify the frequency in iterations at which the mesh will be updated within a time step.

Motion Relaxation
allows you to set a value (within the range of 0 to 1) for ω in Equation 10.29 (p. 711), which defines the relaxation of the motion (that is, displacement of the nodes) during the mesh update.

Residual Criteria
allows you to set the relative residual threshold that is used to check the motion convergence.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Contact Detection
contains parameters to be specified for contact detection. See Contact Detection Settings (p. 711) for more information.

Face Zones
allows you to select the face zones that will be involved in contact detection.

Proximity Threshold
allows you to set the threshold below which the user-defined function for contact detection will be invoked.

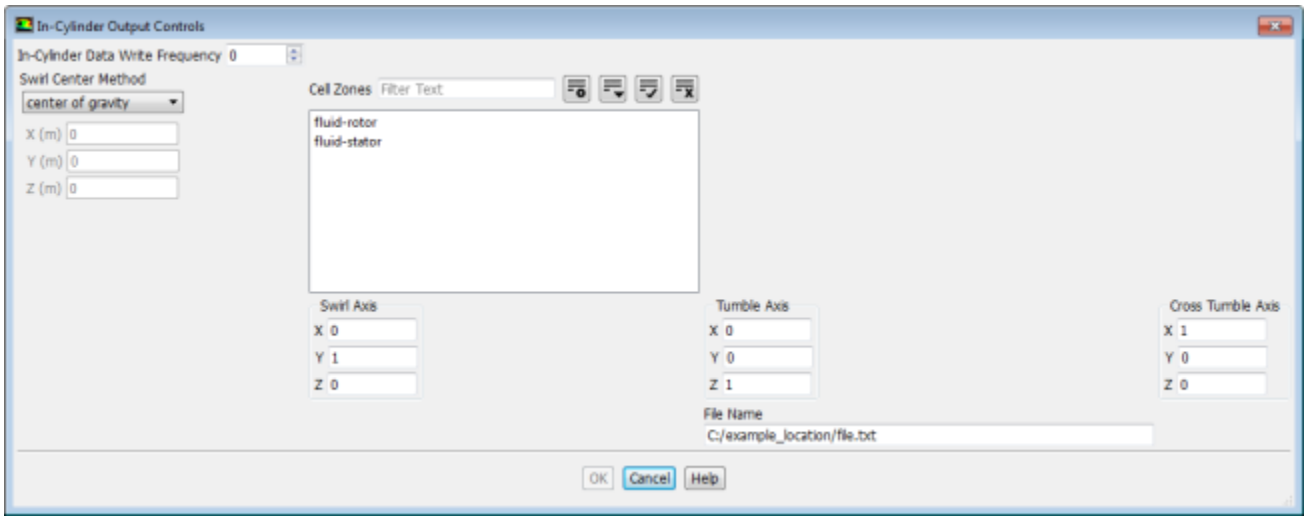
UDF
allows you to specify the user-defined function that will be invoked when contact has been detected.

Flow Control
allows you to indicate whether flow control zones are going to be used in the contact detection process.

Controls...
opens the Flow Controls Dialog Box (p. 2485) where you can make additional flow control settings.

37.10.4. In-Cylinder Output Controls Dialog Box

The **In-Cylinder Output Controls** dialog box contains parameters that control the output for the in-cylinder model. See In-Cylinder Settings (p. 690) for more information.



Controls

In-Cylinder Data Write Frequency
represents an integer entry specifying the interval in number of time-steps. Make sure that a value other than 0 is used for the frequency, in order to allow you to complete your setup.

Swirl Center Method
contains a drop-down list that allows you to select the method to calculate the swirl center. The list contains **center of gravity** and **fixed**, with **center of gravity** being the default value.

center of gravity
calculates the swirl center inside the code and is used as the center of gravity of the chosen cell zones.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

fixed
enables you to specify a swirl center in the **X**, **Y**, and **Z** entry fields below the drop-down list.

In addition to these two options, you can chose to use your own **Compiled** user-defined function to calculate the swirl center. For details on using a dynamic mesh UDF, see the separate Fluent Customization Manual.

Cell Zones
is a list that displays the names of all existing cell zones in the case files. You can select only the zones relevant for the swirl and tumble calculations.

Swirl Axis
specifies the swirl axis with three entries for the directional components. By default, **X**, **Y**, **Z**= **0**, **1**, **0**.

Tumble Axis
specifies the directional components of **Tumble Axis** in **X**, **Y**, **Z** directions. By default, **X**, **Y**, **Z** = **0**, **0**, **1**. This applies only in 3D.

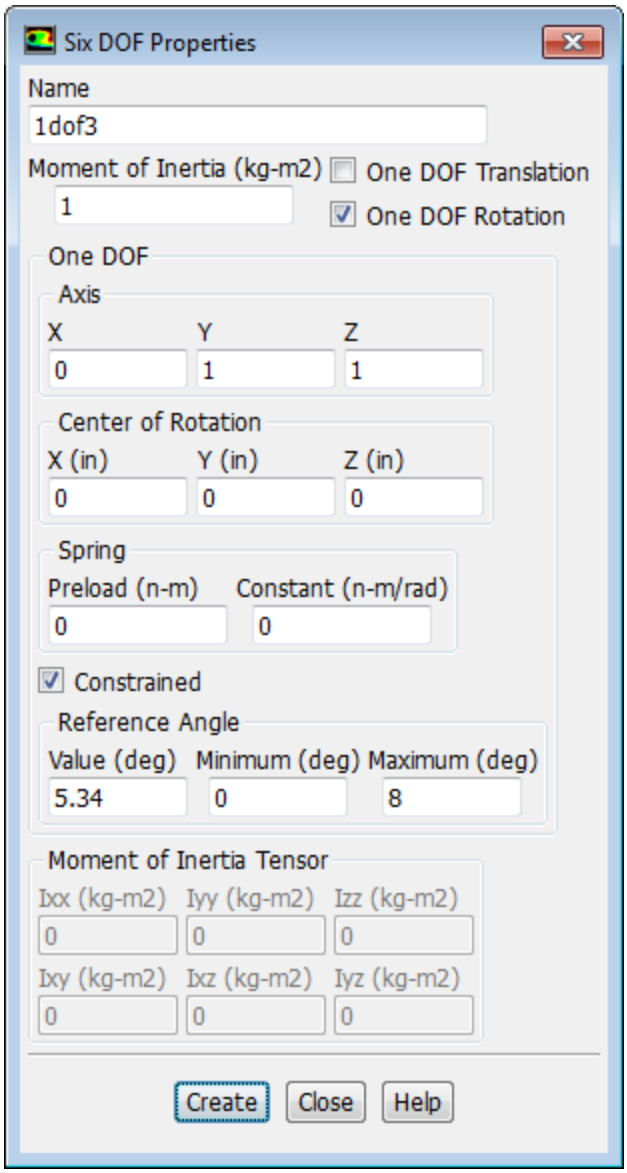
Cross Tumble Axis
specifies the directional components of **Cross Tumble Axis** in **X**, **Y**, **Z** directions. By default, **X**, **Y**, **Z** = **1**, **0**, **0**. This applies only in 3D.

File Name
specifies the name of the **In-Cylinder** output file. By default, the file name contains the name of the case file appended with a `.txt` extension.

37.10.5. Six DOF Properties Dialog Box

The **Six DOF Properties** dialog box allows you to define a set of properties for a rigid body dynamic object that uses the **Six DOF** solver. The set of properties that you create can then be selected in the **Six DOF UDF/Properties** drop-down in the Dynamic Mesh Zones Dialog Box (p. 2490) for the zones that make up the object. See Setting Rigid Body Motion Attributes for the Six DOF Solver (p. 707) and Rigid Body Motion (p. 726) for more information.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

specifies the name of the set of properties, which will be selectable in the **Six DOF UDF/Properties** drop-down in the Dynamic Mesh Zones Dialog Box (p. 2490).

Mass

defines the mass of the rigid body dynamic object that undergoes six degrees of freedom (DOF) motion or one DOF translation.

Moment of Inertia

defines the moment of inertia for the rigid body dynamic object that undergoes one degree of freedom (DOF) rotation.

One DOF Translation

specifies that the dynamic object will only undergo one DOF translation.

One DOF Rotation

specifies that the dynamic object will only undergo one DOF rotation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

One DOF
contains settings that define the one DOF translation or rotation.

Direction
defines the vector along which the object will be allowed to translate.

Axis
defines the components of the axis of rotation. Note that for 2D cases, the axis will be either the positive or negative *z* direction, depending on the sign of the value you enter for **Z**.

Center of Rotation
defines the point about which the object rotates.

Spring
contains settings that define an optional Hooke's law spring that can exert forces / torques on the object. It is a compression / extension spring for translations, and it is a torsion spring for rotations.

Preload
defines a spring load that is constantly applied to the object, in addition to the loading resulting from the spring **Constant**. The **Preload** can be a positive or negative value, and is defined relative to the **Direction** (for translation) or the **Axis** (for rotation).

Constant
defines the spring constant / rate.

Constrained
specifies that the range of motion of the object is limited, as defined by the settings in the **Reference Point** group box (for translation) or the **Reference Angle** group box (for rotation).

Reference Point
contains settings that define the limits of the object's translation when the **Constrained** option is enabled.

Location
assigns a coordinate value to a point located on the object in its initial position; the **Minimum** and **Maximum** coordinates are then defined relative to this value along the **Direction** vector, in order to establish the limits of translation.

Minimum
defines the lower extent of the constrained translation along the **Direction** vector, relative to the **Location**.

Maximum
defines the upper extent of the constrained translation along the **Direction** vector, relative to the **Location**.

Reference Angle
contains settings that define the limits of the object's rotation when the **Constrained** option is enabled.

Value
assigns an angular coordinate value to the orientation of the object in its initial position; the **Minimum** and **Maximum** angles are then defined relative to this orientation about the axis of rotation, in order to establish the limits of rotation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

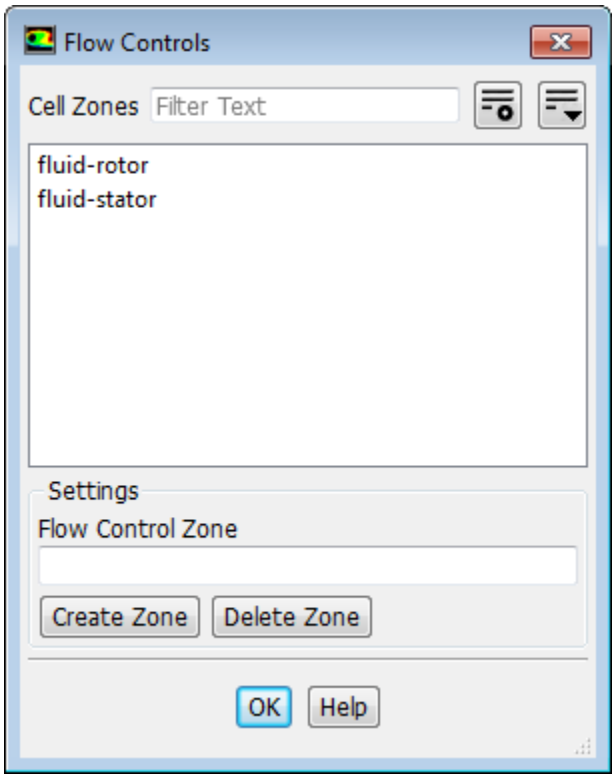
Minimum
defines the lower extent of the constrained rotation about the axis of rotation, relative to the **Value**.

Maximum
defines the upper extent of the constrained rotation about the axis of rotation, relative to the **Value**.

Moment of Inertia Tensor
defines the components of the inertia tensor for the object during six DOF motion. Note that the values are relative to the **Center of Gravity** and **Rigid Body Orientation** settings defined in the Dynamic Mesh Zones Dialog Box (p. 2490); the latter settings allow you to use a coordinate system that is local (rather than global), if that is more convenient.

37.10.6. Flow Controls Dialog Box

The **Flow Controls** dialog box allows you to specify flow control cell zones that are going to be used in the contact detection process. See Contact Detection Settings (p. 711) for more information.



Controls

Cell Zones
is a list that displays the names of all existing cell zones.

Settings
contains flow control zone settings.

Flow Control Zone
allows you to specify a name for the flow control zone.

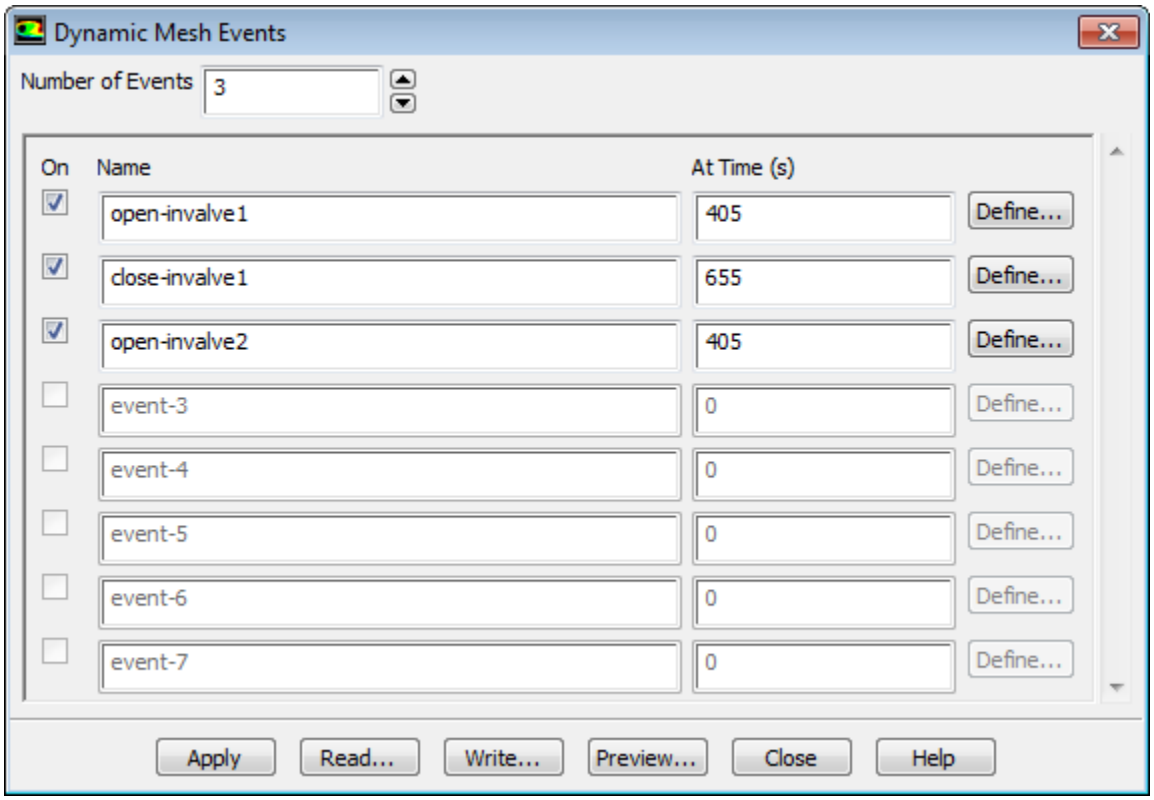
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create Zone
allows you to create a new flow control zone of the specified name.

Delete Zone
allows you to delete the selected flow zone.

37.10.7. Dynamic Mesh Events Dialog Box

The **Dynamic Mesh Events** dialog box is available to control the timing of specific events during the course of the simulation. See Defining Dynamic Mesh Events (p. 713) for details.



Controls

Number of Events
specifies the number of events to be defined.

On
enables the corresponding event.

Name
specifies the name of the event.

At Crank Angle
specifies the angular location of the crank at which the event should occur. This option appears for in-cylinder flows.

At Time
specifies the time (in seconds) at which you want the event to occur. This option appears for non-in-cylinder flows.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Define...
opens the Define Event Dialog Box (p. 2487).

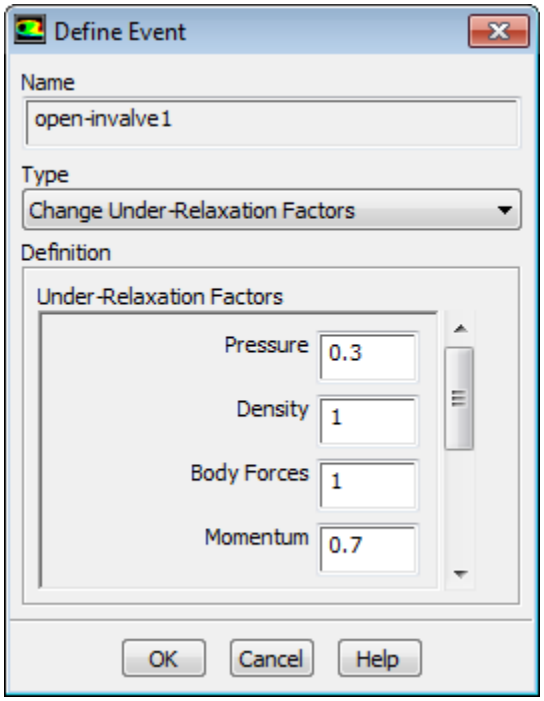
Read...
opens The Select File Dialog Box (p. 19).

Write...
opens The Select File Dialog Box (p. 19).

Preview...
opens the Events Preview Dialog Box (p. 2489).

37.10.8. Define Event Dialog Box

The **Define Event** dialog box allows you to define events.



Controls

Name
contains the name of the event to be defined.

Type
specifies the type of event. You can choose the type of event from the drop-down list. These event types and their definitions are described in Events (p. 716).

Definition
contains the input parameters corresponding to the type of event selected under **Type**.

Zone
contains a selectable list of the zones. The selection specifies the name of the zone(s) to be changed. This item will appear only for the **Change Zone Type** event.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

New Zone Type
specifies the type of zone to which an existing zone must be changed. This item will appear only for the **Change Zone Type** event.

From Zone
specifies the name of the zone from which the boundary condition is to be copied. This item will appear only for the **Copy Zone BC** event.

To Zone(s)
specifies the name of the zone to which the boundary condition is to be copied. This item will appear only for the **Copy Zone BC** event.

Zone(s)
contains a selectable list of the zones. The selection specifies the name of the zone to be activated/de-activated. This item will appear only for the **Activate Cell Zone** and **Deactivate Cell Zone** events.

Interface Name
contains the name of the interface to be created or deleted. This item will appear only for the **Create Sliding Interface** and **Delete Sliding Interface** events.

Interface Zone 1, Interface Zone 2
specifies the two zones on either side of the interface to be created. This item will appear only for the **Create Sliding Interface** event.

Wall 1 Motion, Wall 2 Motion
specifies the dynamic zones whose motion can be copied for the zones specified under **Interface Zone 1** and **Interface Zone 2**. This item will appear only for the **Create Sliding Interface** event.

Attribute
allows you to select the relevant motion attribute. This item will appear only for the **Change Motion Attribute** event.

Status
allows you to enable or disable the motion attribute. This item will appear only for the **Change Motion Attribute** event.

Dynamic Mesh Zones
contains a list of dynamic zones. This item will appear only for the **Change Motion Attribute** event.

Crank Angle Step Size
specifies the new physical time step value in degrees. This item will appear only for the **Change Time Step Size** event.

Base Dynamic Zone
specifies the zone from which the layer of cells is to be created. This item will appear only for the **Insert Boundary Layer** and **Remove Boundary Layer** event.

Side Dynamic Zone
represents the deforming face zone adjacent to the **Base Dynamic Zone** before the layer is inserted. This item will appear only for the **Insert Boundary Layer** event.

Internal Zone 1 Name, Internal Zone 2 Name
specifies the name of the new internal zones.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Time Step Size
allows you to change the time step size of the event. This item will appear only for the **Change Time Step Size** event.

Under-Relaxation Factors
allows you to specify the under-relaxation factors for the selected event. This item will appear only for the **Change Under-Relaxation Factors** event. See Setting Under-Relaxation Factors (p. 1564) for details.

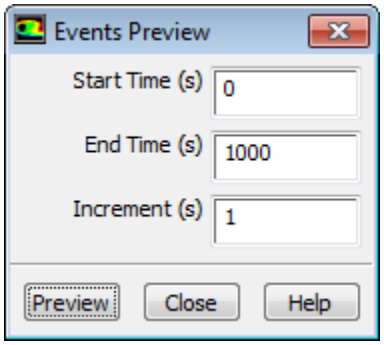
Adjacent Dynamic Face Zone
allows you to select the dynamic face zone adjacent to the location of the cell layer to be inserted (or deleted). You can select the required zone from the drop-down list.

Direction Parameter
is the direction with respect to the selected dynamic face zone a layer of cells is removed or added.

Command
specifies the series of text / Scheme commands or the macro to be executed during the simulation. This item will appear only for the **Execute Command** event.

37.10.9. Events Preview Dialog Box

The **Events Preview** dialog box allows you to play the events to check that they are defined correctly.



Controls

Start Crank Angle
specifies the crank angle at which you want to start the preview (for in-cylinder flows).

Start Time
specifies the time at which you want to start the preview (for non-in-cylinder flows).

End Crank Angle
specifies the crank angle at which you want to end the preview (for in-cylinder flows).

End Time
specifies the time at which you want to end the preview (for non-in-cylinder flows).

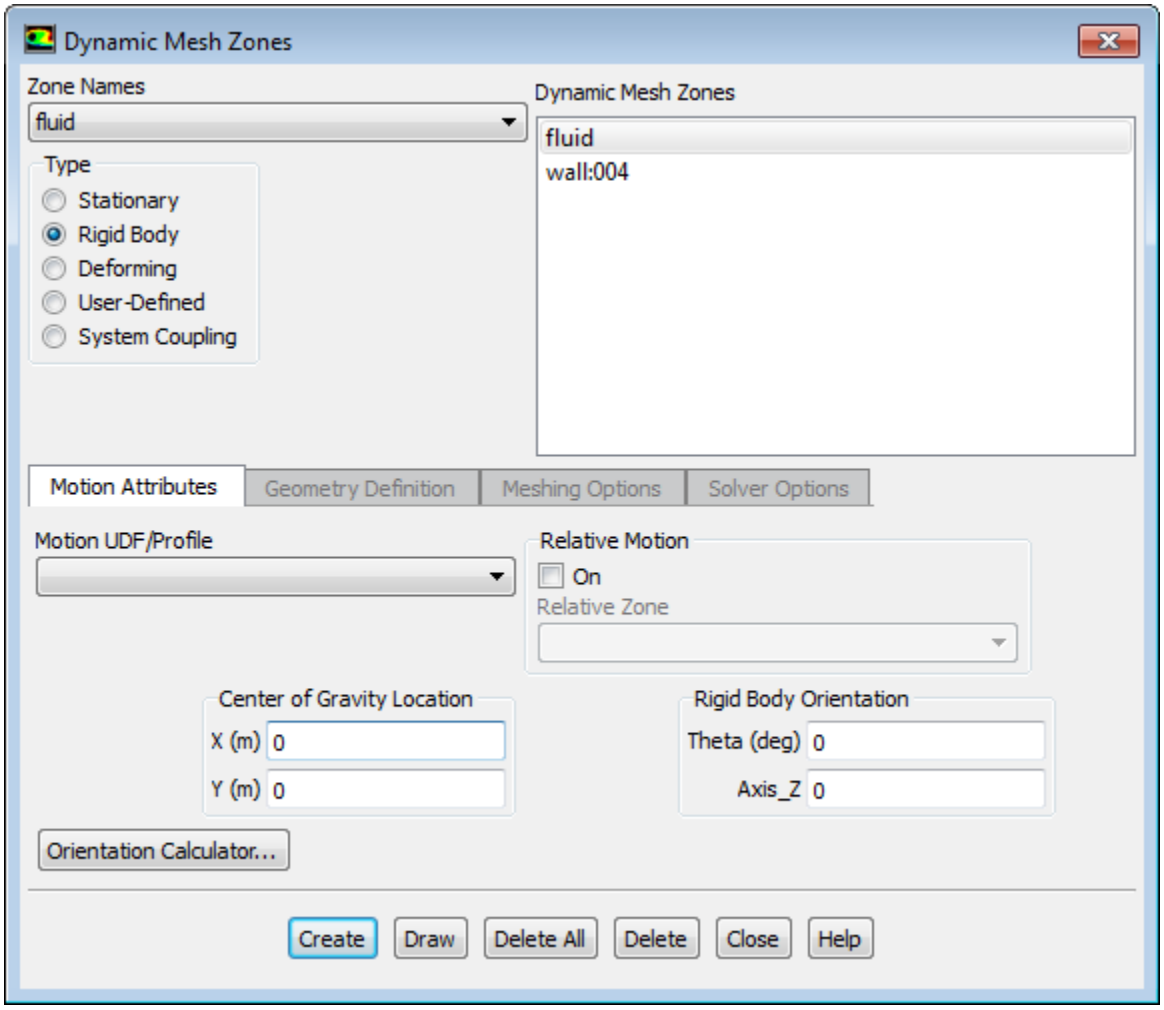
Increment
specifies the size of the step to take during the preview.

Preview
plays back the events at the crank angle specified for each event and reports when each event occurs in the text (console) window.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.10.10. Dynamic Mesh Zones Dialog Box

The **Dynamic Mesh Zones** dialog box allows you to specify the motion of the dynamic zones in your model. See Specifying the Motion of Dynamic Zones (p. 722) for details.



Controls

Zone Names
contains the names of the zones in the model.

Type
contains the types of motion that can be specified for a dynamic zone.

Stationary
explicitly declares the zone as stationary, so that the nodes on this zone are excluded when updating the node positions. See Specifying the Motion of Dynamic Zones (p. 722) for more information.

Rigid Body
specifies the zone as having a rigid-body motion.

Deforming
specifies the zone as deforming.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

User-Defined
enables you to specify a user-defined zone motion.

System Coupling
allows the zone to be involved in a system coupling simulation where the motion is defined by the application that ANSYS Fluent is coupled with on this zone (see the Fluent in Workbench User's Guide and the System Coupling Guide for more details). Once enabled, the **Solver Options** tab can be used to help achieve convergence for system coupling cases where the physics is strongly coupled. If a dynamic mesh zone of the type **System Coupling** exists, and ANSYS Fluent is not involved in a system coupling simulation, then this zone type behaves in the same way as a stationary zone.

Dynamic Mesh Zones
lists all of the dynamic zones in the case.

Motion Attributes
contains parameters to specify the motion attributes for a rigid-body-motion zone and a user-defined-motion zone.

Motion/UDF Profile
allows you to define the motion of the rigid body zone by selecting a profile or user-defined function from the drop-down list. See Profiles (p. 435) and Solid-Body Kinematics (p. 739) for information on profiles, and see the Fluent Customization Manual for information on user-defined functions.

Six DOF UDF/Properties

allows you to define the motion of a rigid body zone by selecting a profile, user-defined function, or set of six DOF properties (created using the Six DOF Properties Dialog Box (p. 2482)) from the drop-down list. This option is available only when **On** is enabled in the **Six DOF** group box.

Six DOF
contains parameters for the **Six DOF** solver.

On
enables the use of the **Six DOF** solver.

Passive
(if enabled) does not take forces and moments on the zone into consideration.

Center of Gravity Location

contains the current values of the coordinates for the location of the center of gravity / rotation of the selected zone. This item will appear only if the motion **Type** is **Rigid Body** and **In-Cylinder** is not enabled in the **Dynamic Mesh** task page.

Relative Motion
allows you to specify the motion of the dynamic zone as relative to another pre-defined rigid body zone. This group box will only appear if the **Type** is **Rigid Body** or **User-Defined**; for the latter, the **Relative Zone** must be a cell zone.

Rigid Body Orientation
contains the current values of the coordinates for the orientation of the rigid body of the selected zone. This item will appear only if the motion **Type** is **Rigid Body** and **In-Cylinder** is not enabled in the **Dynamic Mesh** task page.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Center of Gravity Velocity

specifies the velocity of the center of gravity / rotation with respect to inertia coordinate system. This option is available only if the **Six DOF** solver is the selected model.

Rigid Body Angular Velocity

specifies the angular velocity of the rigid body with respect to inertia coordinate system. This option is available only if the **Six DOF** solver is the selected model.

Lift/Stroke

contains the current value of valve lift or piston stroke, which is automatically updated when you click **Create**. This item will appear only if **In-Cylinder** is enabled in the **Dynamic Mesh** task page.

Valve/Position Axis

specifies the direction of the reference axis of the valves or piston for an in-cylinder problem. This item will appear only if **In-Cylinder** is enabled in the **Dynamic Mesh** task page.

Mesh Motion UDF

allows you to select the user-defined function that defines the geometry and motion of the zone. This item will appear only if the motion **Type** is **User-Defined**.

Exclude Mesh Motion in Boundary Conditions

allows you to specify that the boundary mesh motion should not be included in the physical boundary conditions of that zone. This option is only available for non-periodic boundary zones.

Geometry Definition

contains parameters to define a deforming zone.

Definition

allows you to select a geometry definition to project nodes of the deforming zone on. Available options are **faceted**, **plane**, **cylinder**, and **user-defined**.

Feature Detection

allows you to preserve features for 3D cases on boundary zones of type **Deforming** or **User-Defined**. Such features may be at the juncture of different boundary zones, or they may be on a single non-planar boundary zone.

Include Features

enables the detection and preservation of features of a particular angular range.

Feature Angle

defines a threshold value in degrees. For details on how this angle is defined, see Feature Detection (p. 690).

Point on Plane

allows you to specify the position of a point on the plane. This item will appear only when you select **plane** for the **Definition**.

Plane Normal

allows you to specify the direction of the plane normal. This item will appear only when you select **plane** for the **Definition**.

Cylinder Radius

allows you to specify the radius of the cylinder. This item will appear only when you select **cylinder** for the **Definition**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cylinder Origin
allows you to specify the location of the cylinder origin. This item will appear only when you select **cylinder** for the **Definition**.

Cylinder Axis
allows you to specify the direction of the cylinder axis. This item will appear only when you select **cylinder** for the **Definition**.

Geometry UDF
allows you to select a user-defined function for a geometry **Definition**.

Meshing Options
contains parameters for various meshing options.

Adjacent Zone
contains the name of the adjacent zone that is involved in local remeshing or dynamic layering. This item will appear only if the zone type is **Stationary**, **Rigid Body**, or **User-Defined**, and if the zone is not the boundary of a CutCell dynamic cell zone.

Cell Height
allows you to specify the ideal height of the adjacent cells as either a constant value or a compiled user-defined function. This item will appear only if the motion **Type** is **Stationary**, **Rigid Body**, or **User-Defined**, and if the zone is not the boundary of a CutCell dynamic cell zone.

Deform Adjacent Boundary Layer with Zone
enables the smoothing of an adjacent boundary layer mesh. This option is not available when **Stationary** or **Deforming** is selected in the **Type** list.

Methods
contains options to specify the mesh update method(s) and controls on a zone by zone basis. These options and parameters will override those that you specified globally in the **Dynamic Mesh** task page. This item will appear only if the motion **Type** is **Deforming**.

Smoothing
enables the smoothing mesh update method.

Layering
enables the layering mesh update method (valid for a prismatic cell zone only). This item will appear only if the zone you are defining is a cell (fluid) zone.

Remeshing
enables the remeshing mesh update method.

Smoothing Elements
allows you to specify the element types for which spring-based smoothing is used. This item will only be available if you have selected a cell zone and if spring-based smoothing has been enabled.

Global Settings
specifies that the zonal settings are the same as set globally in the **Mesh Method Settings** dialog box (see Applicability of the Spring-Based Smoothing Method (p. 649) or Mesh Method Settings Dialog Box (p. 2474)).

Tet in Tet Zones
(3D only) specifies that this zone will only get smoothed if it consists entirely of tetrahedral elements.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tri in Tri Zones
(2D only) specifies that this zone will only get smoothed if it consists entirely of triangular elements.

Tet in Mixed Zones
specifies that all simplex elements in this zone should get smoothed. All other elements in a mixed element zone will not undergo smoothing.

All
specifies that all element types in this zone will get smoothed.

Zone Parameters
contains a set of remeshing criteria that, if revised (with reasonable values), will be used for the dynamic zone rather than those you specified globally in the **Remeshing** tab of the **Mesh Method Settings** dialog box. Note that this group box is not available if the **CutCell** option in the **Remeshing Options** group box is enabled.

Minimum Length Scale
allows you to specify the minimum length scale for the zone. This item will appear only if the zone you are defining is a cell (fluid) zone.

Maximum Length Scale
allows you to specify the maximum length for the zone. This item will appear only if the zone you are defining is a cell (fluid) zone.

Maximum Skewness
allows you to specify the desired value for maximum skewness.

Zone Scale Info...
opens the Zone Scale Info Dialog Box (p. 2499), in which you can view the statistics of the mesh, such as minimum, maximum, and average length scale values, as well as the maximum skewness.

CutCell Zone Parameters
contains a set of global parameters used for CutCell zone remeshing. This group box will only appear if the zone selected in the **Dynamic Mesh Zones** list is a cell zone, and if the **CutCell** option is enabled in the **Remeshing Options** group box.

Maximum Mesh Size
allows you to specify the maximum mesh size of the Cartesian CutCell mesh.

Growth Rate
allows you to specify the default growth rate used for mesh refinement by size functions. This growth rate is used during remeshing by the mesh-based size functions of all boundaries of the CutCell cell zone that are not defined as dynamic mesh zones.

Minimum Orthogonal Quality
allows you to specify the minimum allowable orthogonal quality for the cell zone. CutCell remeshing will occur if the orthogonal quality for any cell in the cell zone drops below this value.

Remeshing Interval
specifies the interval of time steps at which the cell zone remeshing will take place, even if the mesh quality has not deteriorated below the value entered for **Minimum Orthogonal Quality**. Enter a value of `0` for the **Remeshing Interval** if you want the remeshing to occur only if there is insufficient mesh quality.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Inflation Layers
allows you to enable inflation layers on the CutCell cell zone and specify global inflation parameters.

Settings...
opens the Inflation Settings Dialog Box (p. 2498), which allows you to enter the global inflation parameters used by default on all boundary zones.

Boundary Zones Info...
opens the CutCell Boundary Zones Info Dialog Box (p. 2498), which lists all of the boundary zones for the CutCell dynamic cell zone, along with the remeshing and inflation layer parameters.

Zone Scale Info...
opens the Zone Scale Info Dialog Box (p. 2499), in which you can view the statistics of the mesh, such as minimum, maximum, and average length scale values, as well as the maximum skewness.

CutCell Boundary Zone Parameters
contains a set of local parameters used for CutCell zone remeshing. This group box will only appear if the zone selected in the **Dynamic Mesh Zones** list is a boundary of a CutCell dynamic cell zone.

Maximum Mesh Size
allows you to specify the maximum mesh size used by the soft size function. If a value of `0` is entered, a mesh-based size function is used to locally refine the Cartesian mesh near the boundary.

Growth Rate
allows you to specify the local growth rate used by the size function.

Zonal Inflation Layer Control
allows you to enable local inflation layer control and specify local inflation parameters. This option is only available if **Inflation Layers** has been enabled for the CutCell dynamic cell zone. If **Zonal Inflation Layer Control** is not enabled, then the global inflation parameters specified for the cell zone are used.

Settings...
opens the Inflation Settings Dialog Box (p. 2498), which allows you to enter local inflation parameters for the CutCell boundary zone.

Zone Scale Info...
opens the Zone Scale Info Dialog Box (p. 2499), in which you can view the statistics of the mesh, such as minimum, maximum, and average length scale values, as well as the maximum skewness.

Smoothing Methods
contains two methods of smoothing. This option is only available if spring-based smoothing has been selected in the **Mesh Method Settings** dialog box.

Spring
enables spring-based smoothing method.

Laplace
enables Laplacian smoothing method.

Remeshing Methods
contains two methods of remeshing. This group box will appear only when the motion **Type** is **Deforming**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Region
enables region-based remeshing method.

Local
enables local remeshing method.

Remeshing Method
provides settings for local face remeshing of the dynamic zone. This group box will appear only for boundary zones, when the motion **Type** is **User-Defined**.

Local
enables local face remeshing for the dynamic zone. This option is only available when **Local Face** is enabled in the **Remeshing** tab of the Mesh Method Settings Dialog Box (p. 2474).

Maximum Skewness
is a field that, if revised (with a reasonable value), sets a maximum skewness for the local face remeshing of the dynamic zone that is different than the global value set in the **Remeshing** tab of the Mesh Method Settings Dialog Box (p. 2474).

Remeshing Options
contains one method of remeshing.

CutCell
enables the CutCell zone remeshing method. This option is only available when the **CutCell Zone** is enabled in the **Mesh Method Settings** dialog box, and if the zone selected in the **Dynamic Mesh Zones** list of the **Dynamic Mesh Zones** dialog box is a cell zone.

Solver Options
contains parameters for various solver options for system coupling and six DOF dynamic mesh zones. See System Coupling Motion (p. 738) for more information.

Solution Stabilization
enables/disables solution stabilization settings for system coupling and six DOF dynamic mesh zones.

Stabilization Parameters
contains options for solution stabilization settings.

Scale Factor
allows you to set the scale factor used by the selected **Method** for convergence stabilization.

Method
Stabilization is achieved through a boundary source coefficient introduced in the continuity equation. It is designed to improve the diagonal dominance of the matrix system in the cells adjacent to system coupling boundaries. Two methods for this boundary source coefficient are available:

volume-based
(default) specifies that the diagonal entry of the linear matrix system corresponding to the discretized continuity equation is re-scaled using the cell volume and a scale factor, according to Equation 10.32 (p. 739).

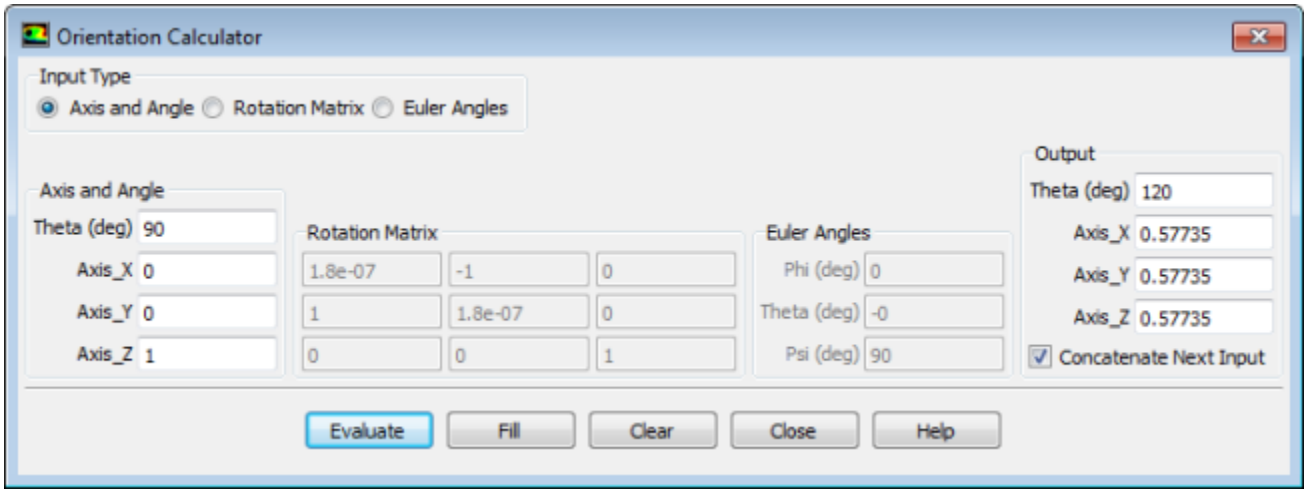
coefficient-based
specifies that the diagonal entry of the linear matrix system corresponding to the discretized continuity equation is directly re-scaled by a scale factor, according to Equation 10.33 (p. 739).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.10.11. Orientation Calculator Dialog Box

The **Orientation Calculator** dialog box allows you to convert rotation matrices and Euler angles to axis and angle format. It also allows you to concatenate rotations to determine a rigid body's final orientation.

Figure 37.1: Orientation Calculator Dialog Box



Controls

Input Type
allows you to select which method you will use to enter your rigid body orientation values. For additional information about these input types, see Rigid Body Motion (p. 726).

Axis and Angle
allows you to enter the angle of rotation and the axis about which the rigid body is rotating.

Rotation Matrix
allows you to enter the orientation of the rigid body as a rotation matrix.

Euler Angles
allows you to enter the orientation of the rigid body as a set of Euler Angles. For an explanation of which Euler angles method Fluent uses, see Rigid Body Motion (p. 726).

Output
displays the angle of rotation and the axis about which the rigid body is rotating after you click **Evaluate**.

Note that if you enter a single orientation in **Axis and Angle** and click **Evaluate**, the values shown in **Output** will be the same as you entered in **Axis and Angle**

Concatenate Next Input
concatenates the next input entered (after clicking **Evaluate**) to the values displayed in **Output**

Evaluate
converts the specified input values into angle and axis format and displays the results in **Output**.

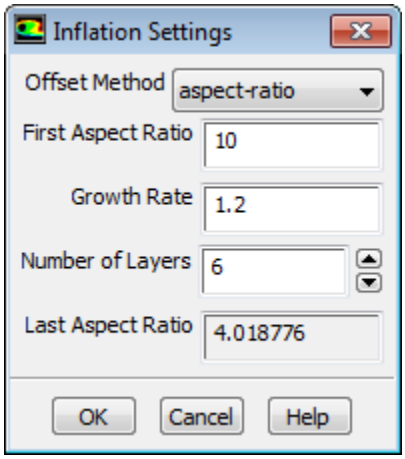
Note that the calculator also converts the entered values into the other two "grayed-out" input formats.

Fill
enters the values displayed in **Output** into the **Rigid Body Orientation** field of the **Dynamic Mesh Zones** dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.10.12. Inflation Settings Dialog Box

The **Inflation Settings** dialog box allows you to enter inflation layer parameters for a dynamic zone.



Controls

Offset Method
allows you to select how the first layer height is evaluated. The **constant** type will use the same first height for all elements. The **aspect-ratio** type will evaluate local first heights such that the aspect ratio (base size divided by the element height) of the resulting elements is constant.

First Aspect Ratio
allows you to specify the aspect ratio of the first element if the **Offset Method** is set to **aspect-ratio**.

First Height
allows you to specify the height of the first element if the **Offset Method** is set to **constant**.

Growth Rate
allows you to specify the geometric growth rate of the inflation element heights.

Number of Layers
allows you to specify the number of elements in the inflation layer.

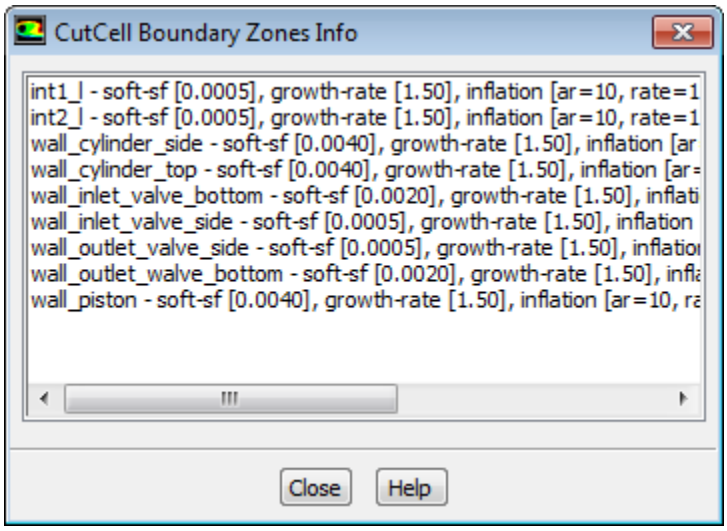
Last Aspect Ratio
displays the aspect ratio of the last element for the given set of parameters if the **Offset Method** is set to **aspect-ratio**.

Total Height
displays the overall thickness of the inflation layer if the **Offset Method** is set to **constant**.

37.10.13. CutCell Boundary Zones Info Dialog Box

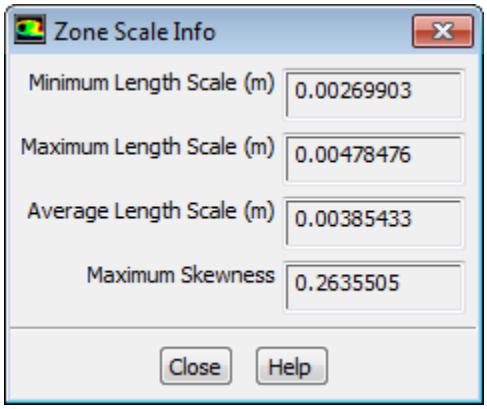
The **CutCell Boundary Zones Info** dialog box displays a list of all of the boundary zones for the CutCell dynamic cell zone. This list also provides the remeshing parameters for the zones, including size function type, the maximum mesh size, the growth rate used to refine the mesh at all of the CutCell boundaries. The inflation layer settings used on all boundaries are also shown.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



37.10.14. Zone Scale Info Dialog Box

The **Zone Scale Info** dialog box allows you to inspect the values of minimum and maximum cell area or volume, and maximum cell skewness in a zone.



Controls

Maximum Length Scale
displays the maximum cell length in the zone.

Minimum Length Scale
displays the minimum cell length in the zone.

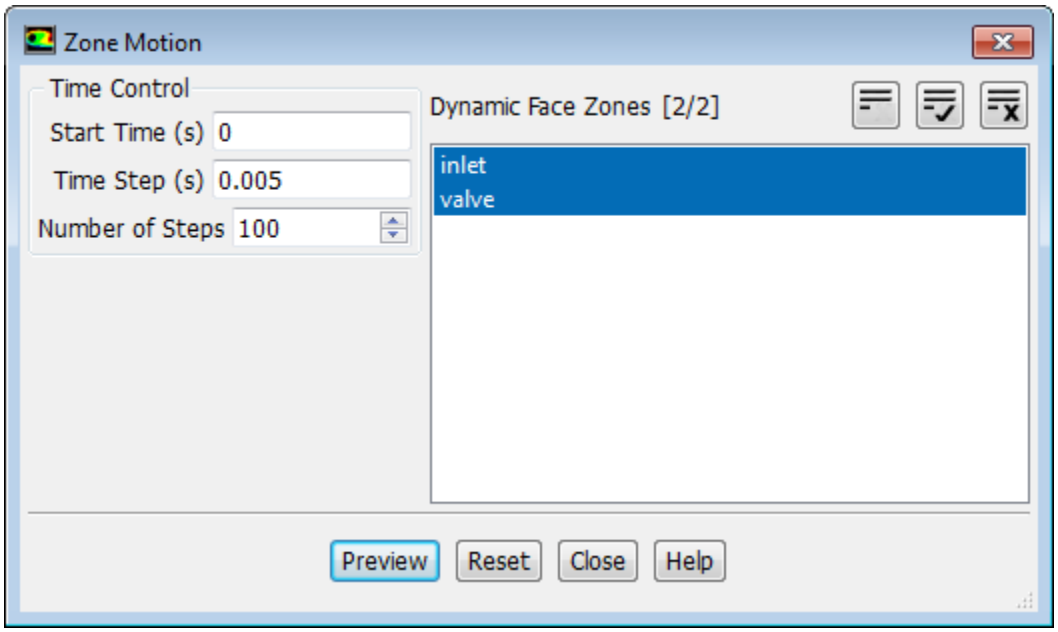
Average Length Scale
displays the average cell length in the zone.

Maximum Skewness
displays the maximum cell skewness in the zone.

37.10.15. Zone Motion Dialog Box

The **Zone Motion** dialog box will display the motion of zones specified with **Rigid Body** or **User-Defined** motion. See Previewing the Dynamic Mesh (p. 742) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Time Control
contains controls to specify the time intervals at which to display the motion.

Start Time (s)
specifies the time from which to start the zone motion preview.

Time Step (s)
specifies time step size for zone motion preview.

Number of Steps
specifies number of time steps for zone motion preview.

Dynamic Face Zones
allows you to select the dynamic face zones to preview. Only **User-Defined** or **Rigid Body** zones are available.

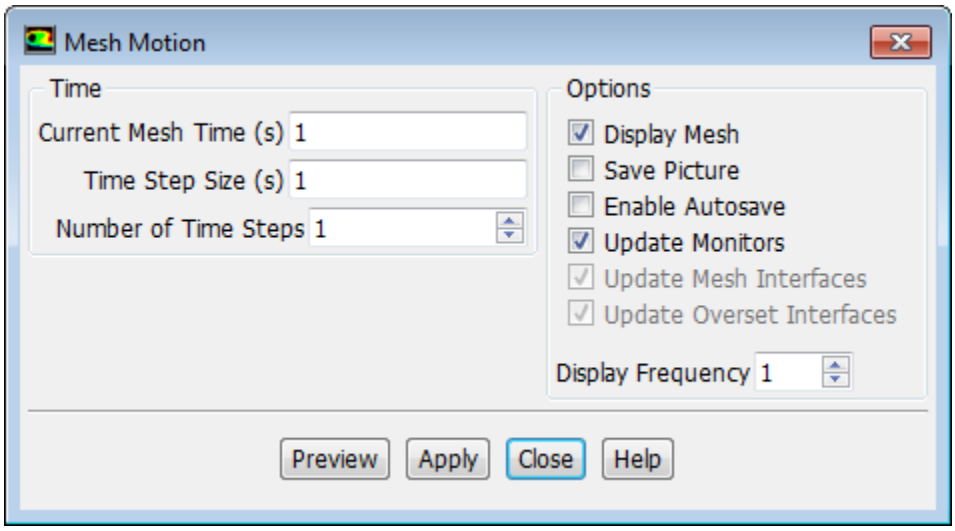
Preview
previews the zone motion.

Reset
resets the zone display and the dialog box inputs.

37.10.16. Mesh Motion Dialog Box

The **Mesh Motion** dialog box allows you to preview the dynamic mesh as it changes with time before you start your simulation. See Previewing the Dynamic Mesh (p. 742) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Time
contains the parameters to specify the time interval at which to update the mesh.

Current Mesh Time
displays the current time after the dynamic mesh has been advanced the specified number of steps.

Time Step Size
specifies the size of each time step.

Number of Time Steps
specifies the number of time steps.

Options
contains options to view the updated mesh.

Display Mesh
displays the mesh.

Save Picture
opens the Save Picture Dialog Box (p. 2568), allowing you to save a picture file of the mesh each time ANSYS Fluent updates it during the mesh preview.

Enable Autosave
opens the Autosave Case During Mesh Motion Preview Dialog Box (p. 2502), allowing you to save the case and data files with the specified name and frequency. See Automatic Saving of Case and Data Files (p. 48) for details.

Update Mesh Interfaces
allows you to update the interface at every time step.

Update Monitors
allows you to disable the processing of monitors and computation activities during mesh motion preview.

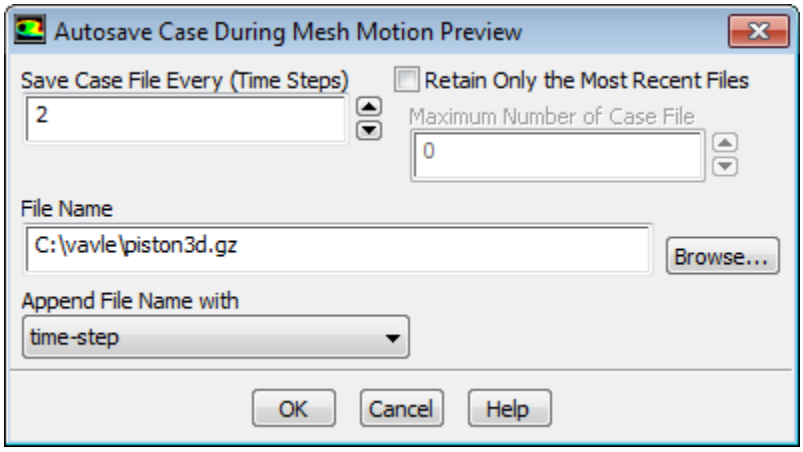
Display Frequency
displays the frequency at which ANSYS Fluent will update the mesh display.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Preview
allows you to preview the motion of the selected zones in the graphics window.

37.10.17. Autosave Case During Mesh Motion Preview Dialog Box

The **Autosave Case During Mesh Motion Preview** dialog box allows you to automatically save your case during the dynamic mesh preview as it changes with time, while running your simulation. See Automatic Saving of Case and Data Files (p. 48) for details about the inputs.



Controls

Save Case File Every
contains the parameters to specify the time interval at which to update the mesh.

Retain Only the Most Recent Files
allows you to restrict the number of files saved by ANSYS Fluent if you have limited disk space. When this option is enabled, you can enter the appropriate value in the **Maximum Number of Data Files** field.

> **Note**
>
> Only the associated case files are retained when using this option.

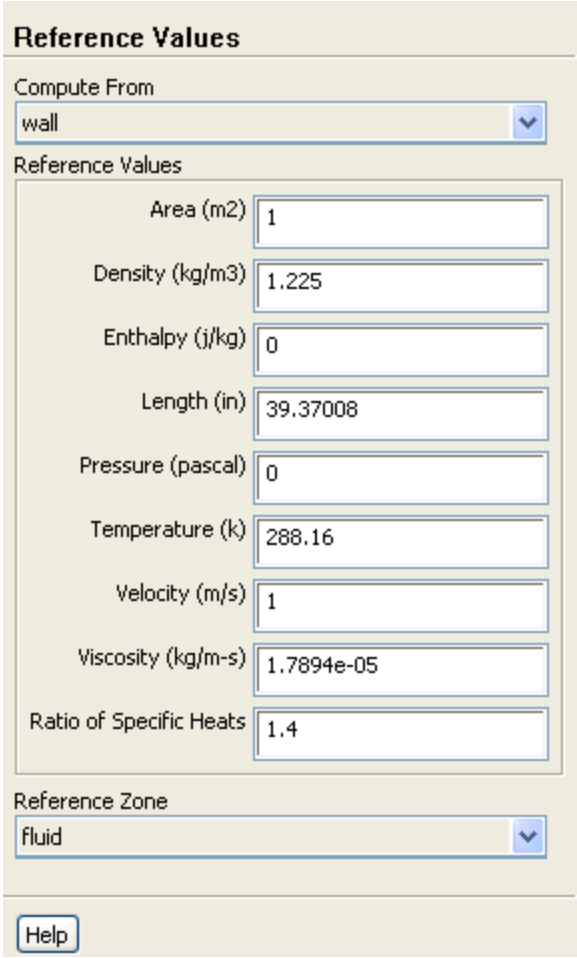
Maximum Number of Data Files
specifies the maximum number of data files that can be saved at any instance. If you have constraints on the disk space, you can restrict the number of files to be saved using this field. After saving the specified number of files, ANSYS Fluent will overwrite the earliest existing file. The default value for this field is zero, which saves all the files.

Append File Name with
allows you to select **flow-time** or **time-step** to be appended to the file name. This option is available only for unsteady-state calculations. The default selection is **flow-time**.

37.11. Reference Values Task Page

The **Reference Values** task page allows you to set the reference quantities used for computing normalized flow field variables. It also allows you to specify the reference zone for postprocessing relative velocities in moving-zone problems. See Reference Values (p. 1924) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Compute from
contains a drop-down list of the boundary zones. You may select a zone to be used for automatically defining the reference values, but depending on the boundary condition used, all of the reference values may not be set. For example, the reference length and area will not be set by computing the reference values from a boundary condition; you must set these manually.

Reference Values
contains inputs for the reference values.

Area
sets the reference area, which is used to compute the force and moment coefficients.

Depth
sets the reference depth used for computing cell volumes in 2D.

Density
sets the reference density, which is used to compute the reference dynamic pressure.

Enthalpy
sets the reference enthalpy, which is used to determine the total enthalpy change.

Length
sets the reference length, which is used in the computation of the moment coefficient.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pressure
sets the reference pressure, which is used to compute the pressure-related forces and moments and the pressure coefficient.

Temperature
sets the reference temperature, which is used to compute entropy for incompressible flows.

Velocity
sets the reference velocity magnitude, which is used to compute the reference dynamic pressure.

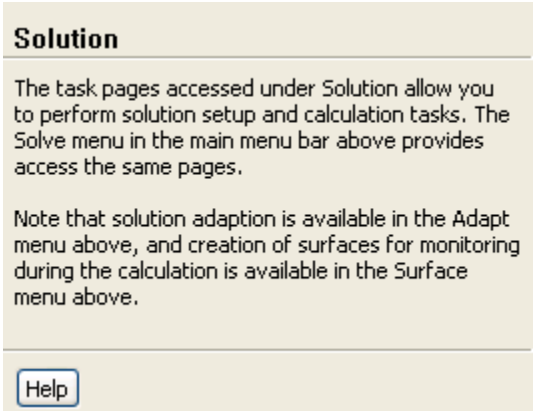
Viscosity
sets the reference kinematic viscosity, which is used in the computation of the boundary Reynolds number.

Ratio of Specific Heats
sets the value of the specific heat ratio, which is used in turbomachinery efficiency calculations.

Reference Zone
contains a drop-down list of all cell zones in the domain. For flows involving multiple moving zones, you must select the reference zone for postprocessing relative velocities and related quantities. See Setting the Reference Zone (p. 1927) for details.

37.12. Solution Task Page

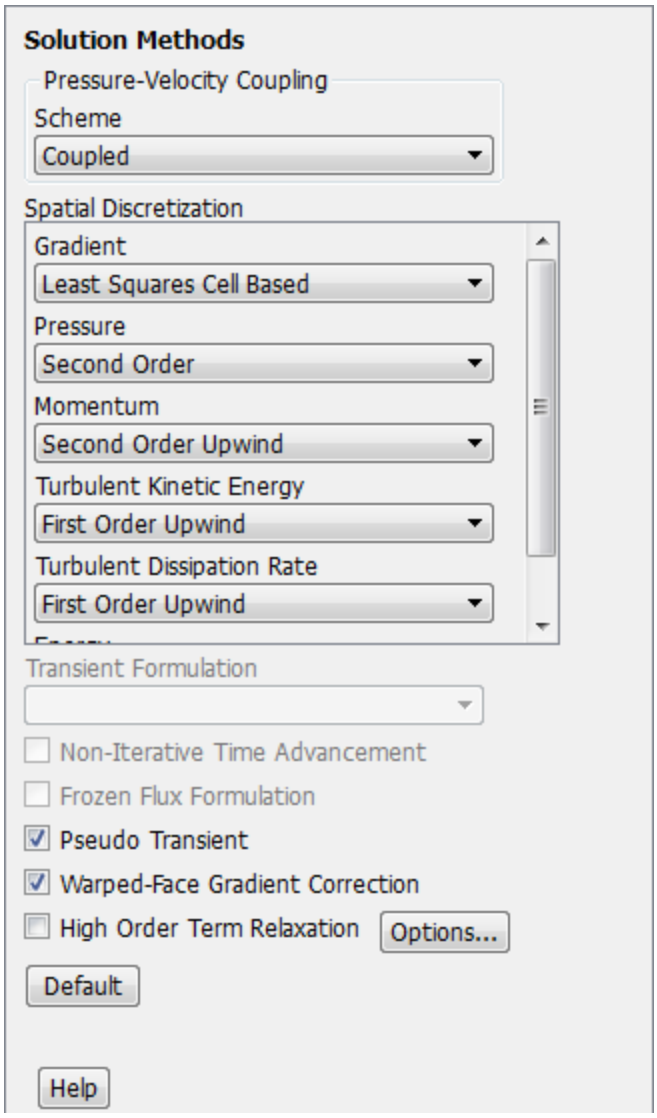
The **Solution** task page introduces you to the main tasks involved in solving your CFD simulation using ANSYS Fluent. For more information about using ANSYS Fluent to solve your CFD simulation, see Using the Solver (p. 1551).



37.13. Solution Methods Task Page

The **Solution Methods** task page allows you to specify various parameters associated with the solution method to be used in the calculation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Formulation

provides a drop-down list of the available types of solver formulations: **Implicit** and **Explicit**.

This item appears only when the density-based solver is used.

Flux Type

provides a drop-down list of the convective flux types: **Roe-FDS**, **AUSM**, and **Low Diffusion Roe-FDS**. Details about each of the flux types can be found in Convective Fluxes in the Theory Guide.

Pressure-Velocity Coupling

contains settings for pressure-velocity coupling schemes.

Scheme

(for the pressure-based solver only) provides a drop-down list of the available pressure-velocity coupling schemes: **SIMPLE**, **SIMPLEC**, **PISO**, and **Coupled**. When the non-iterative time advancement (NITA) scheme is enabled in the **Solution Methods** task page, **Fractional Step** is available for single phase flows, and **Phase Coupled SIMPLE** is available for Eulerian multiphase flows. See Pressure-Velocity Coupling in the *Fluent Theory Guide*, and Choosing the Pressure-Velocity Coupling Method (p. 1561),

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Solution Controls for the Non-Iterative Solver (p. 1566), and Selecting the Pressure-Velocity Coupling Method (p. 1507) for details about these methods.

Skewness Correction
enables the SIMPLEC and PISO skewness correction for highly skewed meshes if the value (number of iterations) is greater than 0. The default value is 0 for SIMPLEC and 1 for PISO.

Neighbor Correction
enables the PISO neighbor correction, which is recommended for transient calculations, if the value (number of iterations) is greater than 0. The default value is 1.

Skewness-Neighbor Coupling
allows for a more economical but a less robust variation of the PISO algorithm.

Coupled with Volume Fraction
couples velocity corrections, shared pressure corrections, and the correction for volume fraction simultaneously. This option is available in the interface after you have selected **Coupled** from the **Scheme** drop-down list for **Pressure-Velocity Coupling**.

Solve N-Phase Volume Fraction Equations
solves all primary and secondary volume fraction equations and scales the resulting volume fractions to satisfy the requirement that they sum to 1. If this is disabled, only the secondary phase volume fractions are solved directly and the primary phase volume fraction is set such that the volume fractions sum to 1.

Spatial Discretization
contains settings that control the spatial discretization of the convection terms in the solution equations. See Choosing the Spatial Discretization Scheme (p. 1554) for details.

Gradient
contains a drop-down list of the options for setting the method of computing the gradient in Equation 21.29 in the Theory Guide: **Green-Gauss Cell Based**; **Green-Gauss Node-Based**, and **Least Squares Cell Based**. See Evaluation of Gradients and Derivatives in the Theory Guide for details.

Pressure
(for the pressure-based solver only) contains a drop-down list of the discretization schemes available for the pressure equation: **Standard**, **PRESTO!**, **Linear**, **Second Order**, and **Body Force Weighted**.

This item appears only when the pressure-based solver is used.

Flow
(for the density-based solvers only) contains a drop-down list of the discretization schemes available for the pressure, momentum, and (if relevant) energy equations: **First Order Upwind**, **Second Order Upwind**, and **Third-Order MUSCL**.

This item appears only when one of the density-based solvers is used.

Momentum, Energy, and so on
are the names of the other convection-diffusion equations being solved. In the drop-down list next to each equation, you can select the **First Order Upwind**, **Second Order Upwind**, **Power Law**, **QUICK**, or **Third-Order MUSCL** discretization scheme for that equation.

If the LES turbulence model is enabled, then you have a choice of selecting **Bounded Central Differencing** or **Central Differencing** to solve the convection-diffusion equations.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If one of the density-based solvers is used, **Momentum** and **Energy** will not appear. For the density-based solvers, the discretization scheme for these equations is selected in the **Flow** drop-down list (described above).

Volume Fraction

is available when the VOF multiphase model is enabled. The discretization schemes that are used when solving volume fraction equations for the VOF explicit scheme are **Geo-Reconstruct**, **Compressive**, **CICSAM**, **Modified HRIC**, and **QUICK**. The discretization schemes that are used when solving volume fraction equations for the VOF implicit scheme are **First Order Upwind**, **Second Order Upwind**, **Compressive**, **Modified HRIC**, **BGM** (steady state only), and **QUICK**. See Interpolation Near the Interface in the Theory Guide for detailed information about these VOF-specific interpolation schemes.

Transient Formulation

contains options for setting different time-dependent solution formulations. This option appears only when **Transient** is enabled under **Time** in the **General** task page. Available options include: **Explicit** (available only for the **Density Based Explicit** solver); **First Order Implicit**; **Second Order Implicit**; and **Bounded Second Order Implicit**. See Performing Time-Dependent Calculations (p. 1608) for details.

Non-Iterative Time Advancement

enables non-iterative time-advancement (NITA) scheme. See Time-Advancement Algorithm in the Theory Guide for details.

Frozen Flux Formulation

enables an option to discretize the convective part of Equation 21.63 in the Theory Guide using the mass flux at the cell faces from the previous time level n. This option is available only for a **Transient** solution. See The Frozen Flux Formulation in the Theory Guide for details.

Pseudo Transient

enables an option to apply the pseudo transient under-relaxation method, which is a form of implicit under-relaxation (see Pseudo Transient Under-Relaxation). This option is available for the pressure-based solver when **Coupled** is selected as the **Pressure-Velocity Coupling** scheme and for the density-based implicit solver. Note that this method can only be used when running a steady-state simulation.

Warped-Face Gradient Correction

enables an adjustment to the gradient discretization method that improves the gradient accuracy for meshes containing high aspect ratios, cells with non-flat faces, and highly deformed cells. By default, the fast, or more computationally efficient mode of the warped-face gradient correction is used. For additional information about the warped-face gradient correction, see Warped-Face Gradient Correction (p. 1674).

Convergence Acceleration For Stretched Meshes

Enable convergence acceleration for stretched meshes to improve the convergence of the implicit density based solver on meshes with high cell stretching (see Convergence Acceleration for Stretched Meshes (CASM) (p. 1573)).

High Order Term Relaxation

enables the relaxation of high order terms to aid in the solution behavior of flow simulations when higher order spatial discretization is used.

Options...

Opens the Relaxation Options Dialog Box (p. 2508).

Set All Species Discretizations Together

enables an option to use the same discretization scheme for all the species rather than setting each of the species individually. Notice that you will no longer see your list of individual species, instead a **Species**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

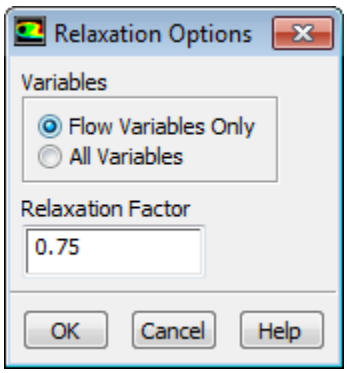
field will appear with the scheme of your choice. Note that this option is available when species transport is enabled.

Default
sets the fields to their default values, as assigned by ANSYS Fluent.

Report Poor Quality Elements
reports statistics on the number and type of cells that ANSYS Fluent has identified as having poor quality.

37.13.1. Relaxation Options Dialog Box

The **Relaxation Options** dialog box allows you to further control the **High Order Term Relaxation** as described in High Order Term Relaxation (HOTR) (p. 1557).



Controls

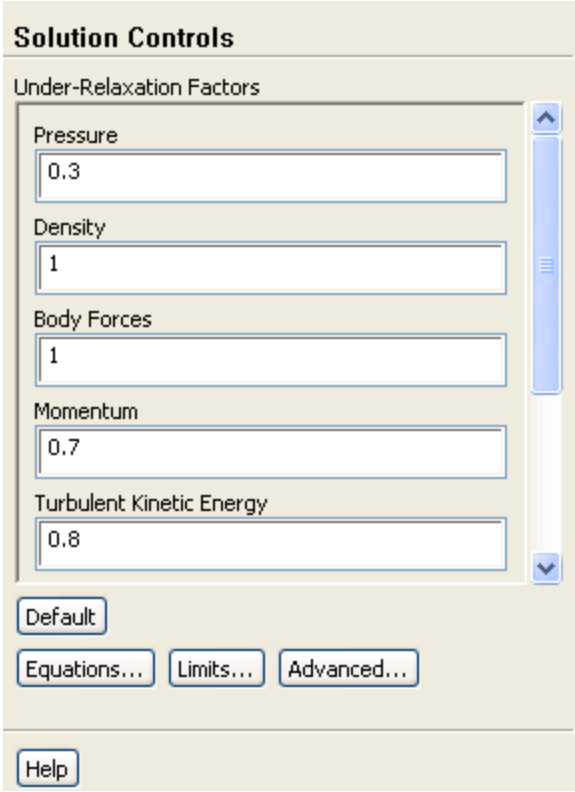
Variables
allows you to select between under-relaxing **All Variables** or only the default flow variables (**Flow Variables Only**.

Relaxation Factor
is by default 0.25 for steady-state cases and 0.75 for transient cases.

37.14. Solution Controls Task Page

The **Solution Controls** task page allows you to set common solution parameters.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Courant Number

sets the fine-grid Courant number (time step factor) when the density-based solver is used. See Changing the Courant Number (p. 1571) for guidelines on setting the Courant number.

When the pressure-based solver is used for time-independent flows, and the **Coupled** pressure-velocity scheme is used, the **Courant Number** is used to stabilize the convergence behavior. See Under-Relaxation of Equations in the Theory Guide for a correlation of the under-relaxation factor and courant number.

Flow Courant Number

sets the Courant number for multiphase flow using the pressure-based coupled solver. The **Flow Courant Number** is used to stabilize the convergence behavior.

Volume Fraction Courant Number

sets the Courant number for multiphase flow using the pressure-based solver is coupled with the volume fraction. The default value is 200 and will appear in the interface when the **Coupled with Volume Fraction** option appears in the **Solution Methods** task page.

Explicit Relaxation Factors

for the **Coupled** scheme defines the explicit relaxation of variables between sub-iterations for momentum and pressure. See Under-Relaxation of Variables in the Theory Guide for information.

Multigrid Levels

specifies the maximum number of coarse levels to be created by the FAS multigrid solver. This item is the same as the **Max Coarse Levels** under **FAS Multigrid Controls** in the **Multigrid** tab of the Advanced Solution Controls Dialog Box (p. 2513), and it appears only when the density-based explicit solver is used.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Residual Smoothing

contains parameters that govern the use of implicit residual smoothing. (See Implicit Residual Smoothing in the Theory Guide and Using Residual Smoothing to Increase the Courant Number (p. 1577) for details.) This section of the task page will appear only when the density-based explicit solver is used.

Iterations

sets the number of iterations of the Jacobi smoother to use. If **Iterations** is 0, then no implicit residual smoothing is performed.

Smoothing Factor

sets the implicit residual smoothing factor. This item will not appear unless **Iterations** is set to a non-zero value.

Under-Relaxation Factors

contains the under-relaxation factors for all equations that are being solved with the pressure-based solver. (See Setting Under-Relaxation Factors (p. 1564) for details.) In the field next to each equation, you can set the under-relaxation factor for that equation.

When one of the density-based solvers is used, **Under-Relaxation Factors** will appear only for the following variables, when they are included in your model: solid (for conjugate heat transfer models), turbulence variables, and viscosity. The solid under-relaxation factor is used to scale the CFL value in the solid region only if the under-relaxation value is set to a value less than 1. When the under-relaxation value is set to 1, then the solver ignores the specified CFL value for the solid and instead uses a very large value of CFL. This is done to speed up the convergence in the solid region. The density-based solvers use a segregated method to solve these equations; all the other equations are solved in a coupled manner, so there are no under-relaxation factors for them.

When **Non-Iterative Time Advancement** is selected for the pressure-based solver, the **Non-Iterative Solver Relaxation Factors** define the explicit relaxation (Under-Relaxation of Variables in the Theory Guide) of variables between sub-iterations and are used to prevent the solution from diverging.

When **Pseudo Transient** is selected (available with the pressure-based coupled solver), the **Pseudo Transient Explicit Relaxation Factors** can be specified (see Setting Pseudo Transient Explicit Relaxation Factors (p. 1602)).

Default

sets the fields to their default values, as assigned by ANSYS Fluent.

Equations...

displays the Equations Dialog Box (p. 2511).

Limits...

displays the Solution Limits Dialog Box (p. 2511).

Advanced...

displays the Advanced Solution Controls Dialog Box (p. 2513).

Set All Species URFs Together

enables an option to use the same under-relaxation factors for all the species rather than setting each of the species individually. Notice that you will no longer see your list of individual species, instead a **Species** field will appear with the specified under-relaxation factor. Note that this option is available when species transport is enabled.

For additional information, see the following sections:

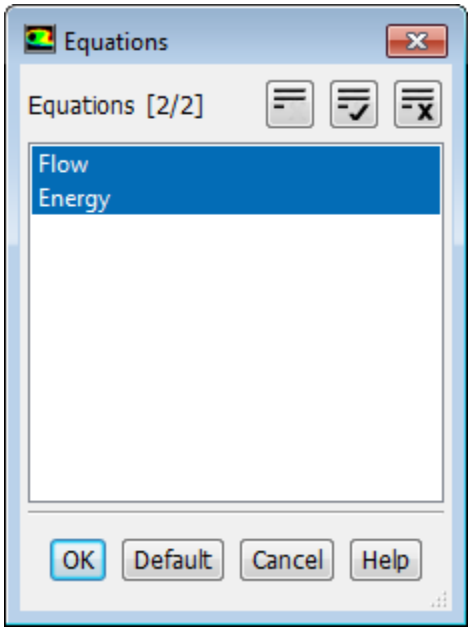
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.14.1. Equations Dialog Box
37.14.2. Solution Limits Dialog Box
37.14.3. Advanced Solution Controls Dialog Box

37.14.1. Equations Dialog Box

The **Equations** dialog box contains a list of the equations being solved for the current model. To temporarily disable solution of an equation, deselect it in this list and click **OK**. To re-enable the calculation for an equation, select it and click **OK**. See Step-by-Step Solution Processes (p. 1669) for details about using this feature in a step-by-step solution process.

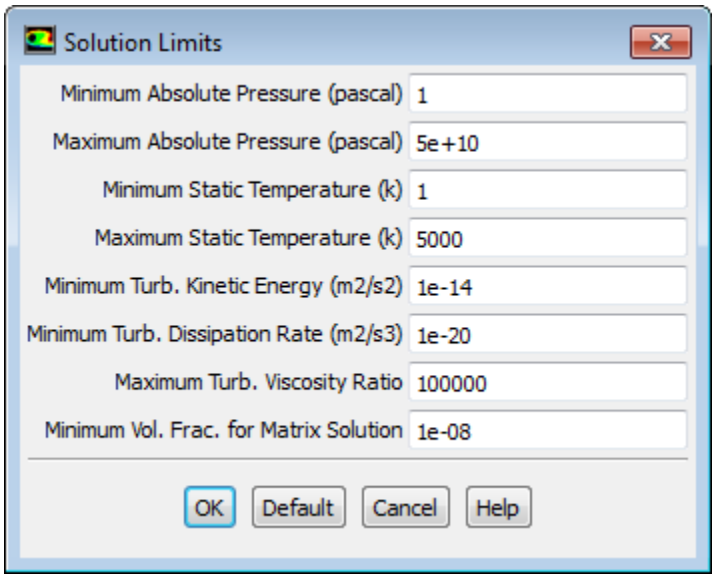
Note that, when one of the density-based solvers is used, **Energy** will not appear as a separate item in the **Equations** list. For the density-based solvers the energy equation is included in the **Flow** category (which also includes the pressure and momentum equations).



37.14.2. Solution Limits Dialog Box

The **Solution Limits** dialog box allows you to improve the stability of the solution. See Setting Solution Limits (p. 1586) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Minimum Absolute Pressure, Maximum Absolute Pressure
set the limiting minimum and maximum allowable values for absolute pressure.

Minimum Static Temperature, Maximum Static Temperature
set the limiting minimum and maximum allowable values for temperature.

Minimum Turb. Kinetic Energy
sets the limiting minimum value of turbulent kinetic energy (k) in the flow field. This parameter appears when one of the k- ε or k- ω models or the RSM is used.

Minimum Turb. Dissipation Rate
sets the limiting minimum value of turbulent dissipation rate (ε) in the flow field. This parameter appears when one of the k- ε models or the RSM is used.

Minimum Spec. Dissipation Rate
sets the limiting minimum value of specific dissipation rate (ω) in the flow field. This parameter appears when one of the k- ω models is used.

Maximum Turb. Viscosity Ratio
sets the limiting maximum allowable value of the ratio of turbulent to laminar viscosity (μ_t / μ). The turbulent viscosity is reduced to the necessary value so as not to exceed the maximum allowable viscosity ratio.

This parameter appears for all turbulent flows.

Minimum Vol. Frac. for Matrix Solution
sets the limiting minimum value for volume fraction in the matrix solution. This field appears for Eulerian multiphase simulations run on a double-precision solver.

Positivity Rate Limit
sets the limiting value for the rate of reduction of temperature. See Adjusting the Positivity Rate Limit (p. 1588) for details.

This item appears only when one of the density-based solvers is used.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Default

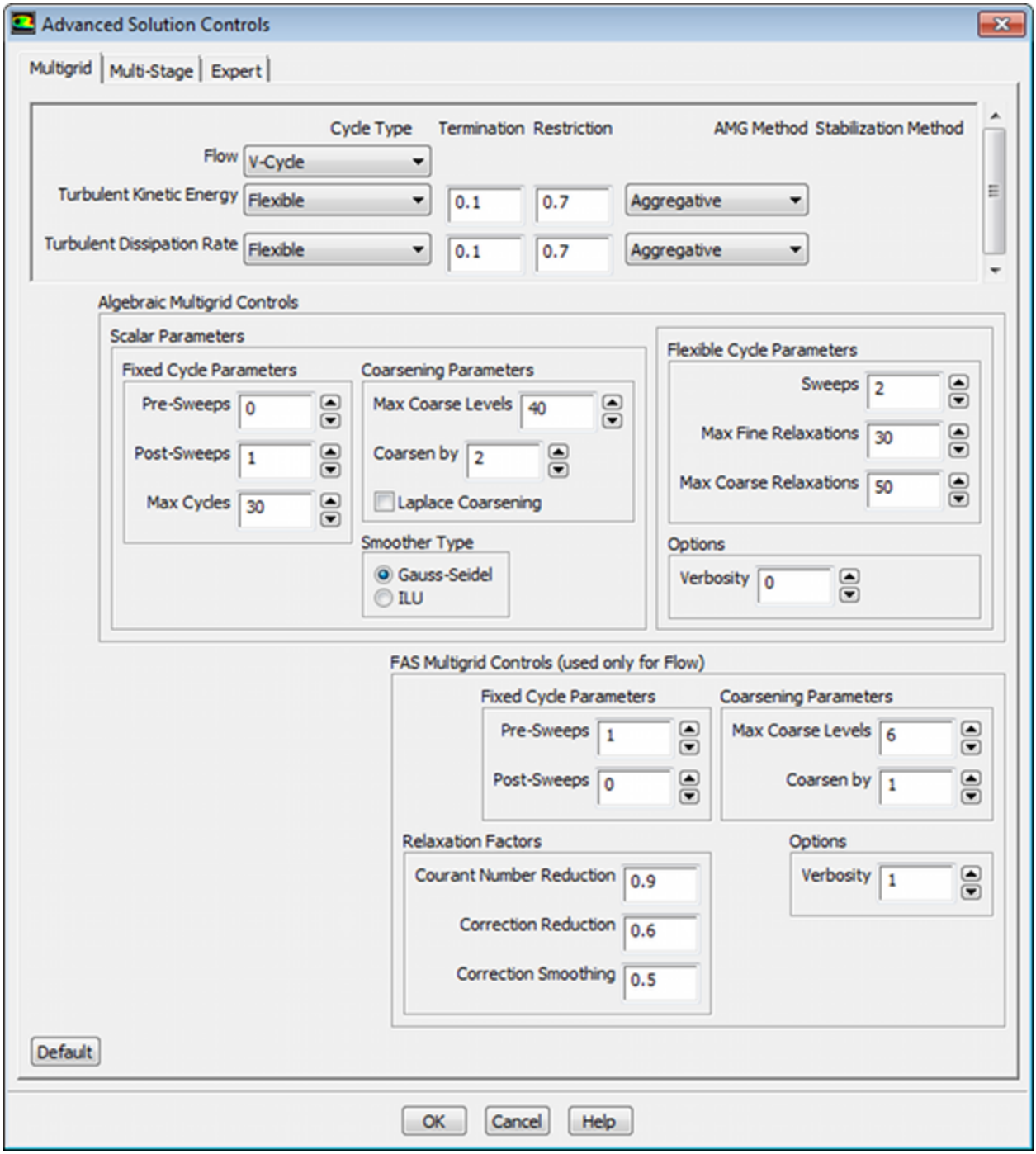
sets the fields to their default values, as assigned by ANSYS Fluent. After execution, the **Default** button becomes the **Reset** button.

Reset

resets the fields to their most recently saved values (for example, the values before **Default** was selected). After execution, the **Reset** button becomes the **Default** button.

37.14.3. Advanced Solution Controls Dialog Box

The **Advanced Solution Controls** dialog box allows you to set parameters related to the multigrid, multi-stage, and non-iterative solvers.



Controls

Multigrid

tab contains parameters related to the multigrid solver. See Modifying Algebraic Multigrid Parameters (p. 1670) and Setting FAS Multigrid Parameters (p. 1583) for details about the items below.

Cycle Type

contains a drop-down list for each equation that is being solved. From this list you can select the multigrid cycle type (**Flexible**, **V-Cycle**, **W-Cycle**, or **F-Cycle**). See Specifying the Multigrid Cycle Type (p. 1578) for details.

Note that, for the density-based solvers, the **Pressure**, **Momentum**, and **Energy** equations will not appear individually. They will instead be grouped together and called **Flow**.

Furthermore, for the density-based explicit solver, the **Cycle Type** choices for the **Flow** equations will be limited to **V-Cycle** and **W-Cycle**, while the choices for **Flow** for the density-based implicit solver will be limited to **V-Cycle** and **F-Cycle**.

Termination

specifies the termination criterion for each equation that is being solved using algebraic multigrid. See Setting the Termination and Residual Reduction Parameters (p. 1578) for details.

Restriction

specifies the residual reduction criterion for each equation that is being solved using the **Flexible** algebraic multigrid cycle. See Setting the Termination and Residual Reduction Parameters (p. 1578) for details. (This item will not appear for an equation that is using a **V-Cycle**, **W-Cycle**, or **F-Cycle**.)

AMG Method

contains the drop-down list to choose between two AMG solvers: aggregative or selective. See Setting the AMG Method and the Stabilization Method (p. 1578) for details.

Aggregative

enables aggregative AMG solver.

Selective

enables selective AMG solver. The selective AMG solver is available only for scalar equations, and is not available in parallel ANSYS Fluent.

Stabilization Method

contains the drop-down list to choose the stabilization method.

BCGSTAB

enables bi-conjugate gradient stabilized method.

RPM

enables the recursive projection method. RPM stabilization is mainly used in conjunction with the coupled pressure-based solver.

CG

enables the conjugate gradient method. CG stabilization is mainly used in conjunction with the segregated pressure-based solver.

Algebraic Multigrid Controls

contains parameters related to the algebraic multigrid solver. See Algebraic Multigrid (AMG) in the Theory Guide and Modifying Algebraic Multigrid Parameters (p. 1670) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scalar and Coupled Parameters
contain parameters that you can set, If you are using the density-based explicit solver or the pressure-based solver with any of the segregated schemes, described in Pressure-Velocity Coupling in the Theory Guide and Choosing the Pressure-Velocity Coupling Method (p. 1561), you will only set **Scalar Parameters**. If you are using the density-based implicit or the pressure-based coupled scheme, described in Coupled Algorithm, then you can set the **Coupled Parameters**.

Fixed Cycle Parameters
contains parameters that control the V, W, and F cycles. You can set the number of **Pre-Sweeps** and **Post-Sweeps**, and the **Max Cycles**. Normally one post-sweep is performed and no pre-sweeps are done. See Fixed Cycle Parameters (p. 1580) for details about using the items below.

Pre-Sweeps
sets the number of sweeps to perform before moving to a coarser level.

Post-Sweeps
sets the number of sweeps to perform after coarser level corrections have been applied.

Max Cycles
sets the maximum number of V, W, or F cycles to be performed. The multigrid solver will continue to solve the set of equations until either the maximum number of cycles has been performed, or the **Termination** criteria are satisfied.

Coarsening Parameters
contains parameters that control the grouping of equations in the algebraic multigrid algorithm. See Coarsening Parameters (p. 1580) for details

Max Coarse Levels
is the maximum number of coarse levels that will be built by the multigrid solver. Sets of coarser simultaneous equations are built until the maximum number of levels has been created, or the coarsest level has only 3 equations. Each level has about half as many unknowns as the previous level, so coarsening until there are only a few nodes left will require about as much total coarse-level coefficient storage as was required on the fine mesh. Reducing the maximum coarse levels will reduce the memory requirements, but may require more iterations to achieve a converged solution. Setting **Max Coarse Levels** to 0 turns off the multigrid solver.

Coarsen by
controls the number of equations on each successively coarser grid level. By default, this parameter is set to 2, indicating that the number of equations on each level will be 1/2 of the number on the previous level. In general, the number of equations on each coarser grid level will be equal to $1/n$ of the number on the previous level, where n is the value set for the **Coarsen by** parameter.

Conservative Coarsening
enables the use of conservative coarsening techniques that can improve parallel performance and/or convergence for some difficult cases. This option is only available under **Coupled Parameters**.

Laplace Coarsening
enables the use of Laplace coefficients when grouping cells for coarsening.

Smoother Type
consist of two types:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Gauss-Seidel
is the simplest smoother type and is recommended when using the pressure-based segregated solver.

ILU
is more CPU intensive, but has better smoothing properties for block-coupled systems such as the pressure-based coupled solver and the density-based implicit solver.

Flexible Cycle Parameters
contains parameters that control the flexible multigrid cycle.

Sweeps
specifies the number of times to apply the smoothing method each time a relaxation is performed.

Max Fine Relaxations
sets the maximum number of relaxations to be performed on the Level 1 grid (fine grid level). This parameter eliminates the possibility that the Gauss-Seidel solver will get "stuck" on the fine grid level, unable to reduce the residuals by the fraction (α) required by the **Termination** criteria.

Max Coarse Relaxations
sets the maximum number of relaxations to be performed on each grid level above Level 1 (that is, the coarse grid levels). This parameter eliminates the possibility that the Gauss-Seidel solver will get "stuck" on a coarse grid level, unable to reduce the residuals on that level by the fraction (α) required by the **Termination** criteria. If the iterative solution on a given grid level is unable to meet the accuracy constraint of the **Termination** criteria, the correction equation will be deemed "converged" when this maximum number of relaxations on that grid level has been performed.

Options
contains additional multigrid parameters.

Verbosity
controls the amount of information that is printed out by the multigrid solver for monitoring purposes. See Setting the Verbosity (p. 1582) for details

FAS Multigrid Controls
contains parameters related to the FAS multigrid solver. See Full-Approximation Storage (FAS) Multigrid in the Theory Guide and Setting FAS Multigrid Parameters (p. 1583) for details. As noted in the title of this dialog box section, the FAS multigrid solver is used only for the **Flow** equations (pressure, momentum, and energy).

This section of the dialog box appears only when the density-based explicit solver is used.

Fixed Cycle Parameters
contains parameters that control the V, W, and F cycles of the FAS multigrid solver.

Pre-Sweeps
sets the number of iterations of the multi-stage solver to be performed on a given grid level before proceeding to a coarser grid level (the value of β_1 described in Multigrid Cycles in the Theory Guide). Typically, this is set to 1.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Post-Sweeps
sets the number of multigrid cycles to be performed on a given grid level before proceeding back up to the finer grid level (the value of β_2 described in Multigrid Cycles in the Theory Guide). A value of 1 results in V-cycle multigrid, and a value of 2 results in W-cycle multigrid.

Coarsening Parameters
contains parameters that control the grouping of cells in the FAS multigrid algorithm.

Max Coarse Levels
sets the maximum number of grid levels to be used in the multigrid process. A value of 0 disables multigrid (no coarse grid levels). If the coarse grids do not already exist, they are created automatically when you start iterating; you cannot create them by clicking the **OK** button. See Turning On FAS Multigrid (p. 1575) for details.

Coarsen by
controls the number of cells in each successively coarser grid level. By default, this parameter is set to 2, indicating that the number of cells on each level will be 1/2 of the number on the previous level. In general, the number of cells on each coarser grid level will be equal to 1/ n of the number on the previous level, where n is the value set for the **Coarsen by** parameter.

Relaxation Factors
are provided to moderate and stabilize the multigrid corrections.

Courant Number Reduction
sets the factor by which to reduce the Courant number for coarse grid levels (that is, every level except the finest). Some reduction of time step (such as the default 0.9) is typically required because the stability limit cannot be determined as precisely on the irregularly shaped coarser grid cells.

Correction Reduction
sets the factor by which to reduce the magnitude of the multigrid corrections transferred from one level to the next finer level. A typical value with $\beta_1 = 1$ is 0.6. If two **Pre-Sweeps** and two **Post-Sweeps** are performed, this value can often be increased to 1.0 (that is, full correction transfer).

Species Correction Reduction
sets the factor by which to reduce the magnitude of the species corrections to stabilize the multigrid calculation. This item appears only when species transport is being modeled.

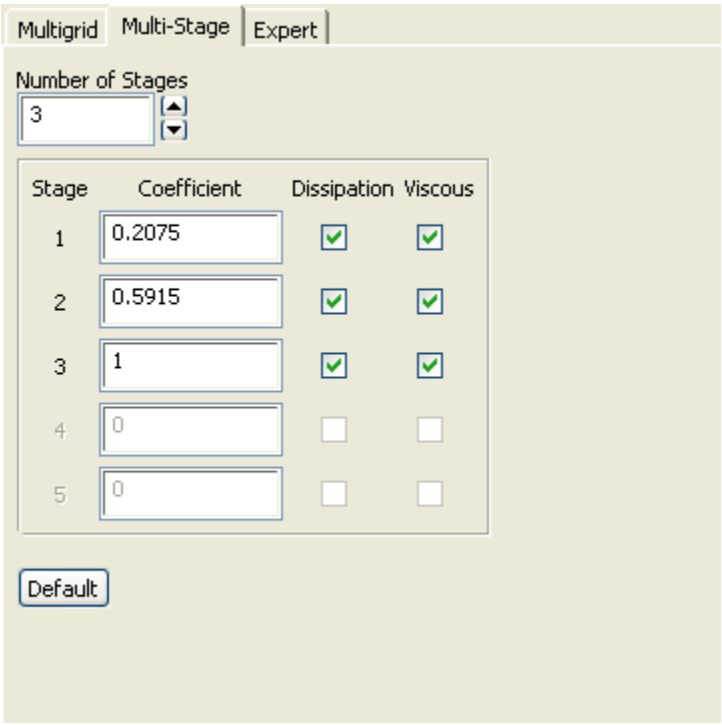
Correction Smoothing
sets the correction smoothing factor used to interpolate corrections from a coarser grid to a finer grid. For multigrid on structured meshes, corrections can be interpolated up to a finer mesh "smoothly" by using, for example, tri-linear interpolation. For unstructured meshes there is no analogous simple, algebraic procedure that can be used to interpolate without introducing substantial high frequency "noise". Instead, the corrections are first interpolated, and then subjected to a smoothing pass. The default **Correction Smoothing** value of 0.5 should be acceptable for all cases; you should not need to change it.

Options
contains additional multigrid parameters.

Verbosity
controls the amount of information that is printed out by the multigrid solver for monitoring purposes.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Multi-Stage
tab allows you to set parameters that govern the operation of the multi-stage solver. It is available only when the density-based explicit solver is used. See Changing the Multi-Stage Scheme (p. 1588) for details about the items below.



Controls

Number of Stages
is the number of stages used in the multi-stage scheme. The default scheme is a 3-stage scheme with coefficients of 0.2075, 0.5915, and 1.0 for the first through third stages, respectively. Although the dialog box limits the maximum number of stages to five, you can define a scheme with an arbitrary number of stages with the `solve/set/multi-stage` text command.

Stage
labels the stage to which the parameters in the other columns apply.

Coefficient
sets the multi-stage coefficient for each stage. Coefficients should be greater than zero and less than one. The final stage should always have a coefficient of 1.

Dissipation
sets the stages for which artificial dissipation is evaluated. If a **Dissipation** box is selected for a particular stage, artificial dissipation will be updated on that stage. If not selected, artificial dissipation will remain "frozen" at the value of the previous stage.

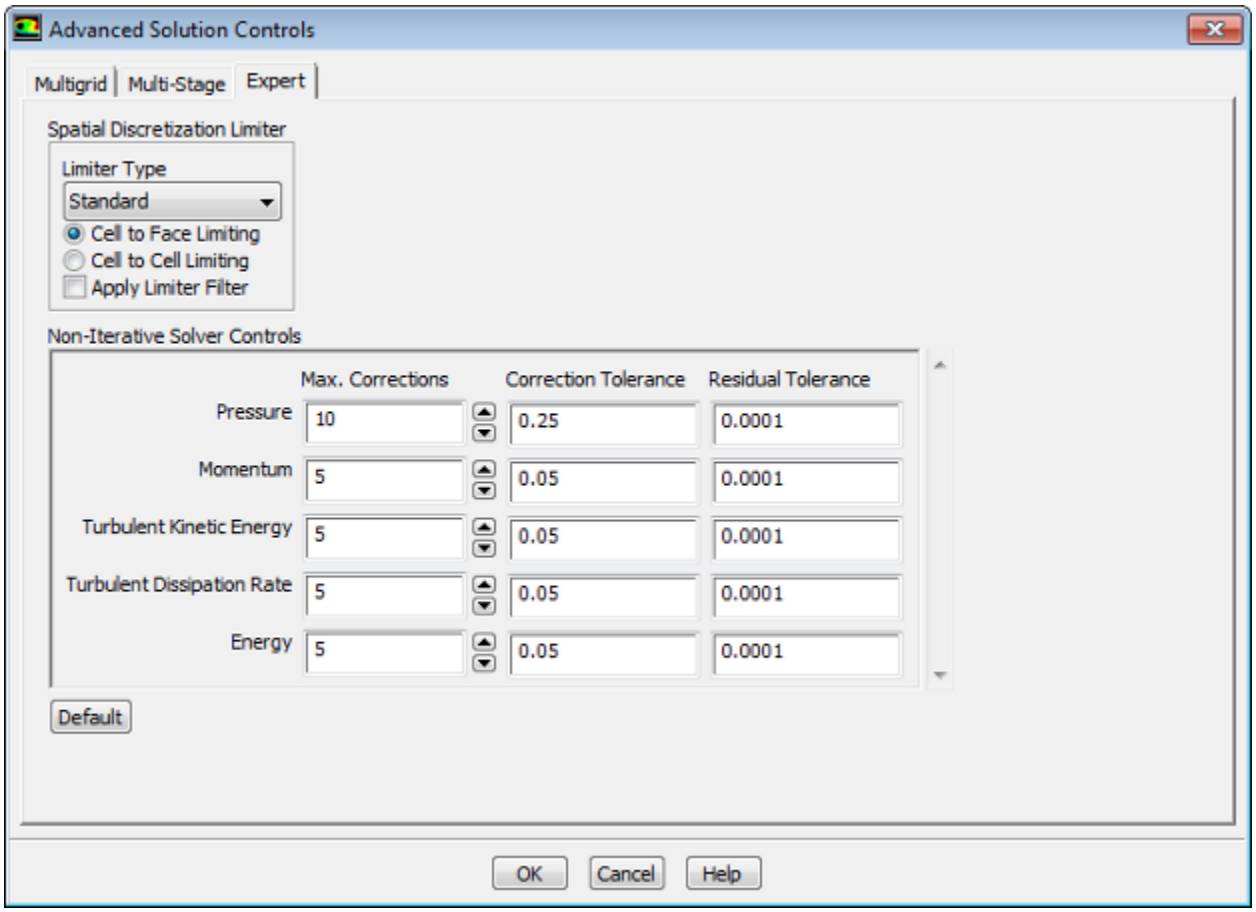
Viscous
sets the stages for which viscous stresses are evaluated. If a **Viscous** box is selected for a particular stage, viscous stresses will be updated on that stage. If not selected, viscous stresses will remain "frozen" at the value of the previous stage. Viscous stresses should always be computed on the first stage, and successive evaluations will increase the "robustness" of the solution process, but will also increase the expense (that is, increase the CPU time per iteration). For steady problems, the final solution is independent of the stages on which viscous stresses are updated.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Default
sets the fields to their default values, as assigned by ANSYS Fluent. After execution, the **Default** button becomes the **Reset** button.

Reset
resets the fields to their most recently saved values (that is, the values before **Default** was selected). After execution, the **Reset** button becomes the **Default** button.

Expert
tab contains specialized parameters for limiting spatial discretization, as well as controls for the non-iterative solver.



Controls

Spatial Discretization Limiter
contains controls for limiting the spatial discretization. See Selecting Gradient Limiters (p. 1590) for more information about limiters.

Limiter Type
selects the type of limiter applied to the spatial discretization: **Standard**, **Multidimensional**, or **Differentiable**.

Cell to Face Limiting
is where the limited value of the reconstruction gradient is determined at the cell face centers. This is the default method.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cell to Cell Limiting
is where the limited value of the reconstruction gradient is determined along a scaled line between two adjacent cell centroids. On an orthogonal mesh (or when the cell-to-cell direction is parallel to face area direction), this method becomes equivalent to the default cell to face method. For smooth field variation, cell to cell limiting may provide less numerical dissipation on meshes with skewed cells.

Apply Limiter Filter
enables the limiter filter. The limiter filter is only available with the **Standard** and **Differentiable** limiter types.

Pseudo Transient Method Usage
allows you to select and modify the parameters for each individual equation. See Setting Solution Controls for the Pseudo Transient Method (p. 1604) for details.

On/Off
enables/disables the equation-specific steady state solution method for a particular equation.

Under-Relaxation Factor
allows you to use the standard steady state method by turning off pseudo transient for that particular equation. Specify the corresponding under-relaxation factor to be employed with a particular equation.

Time Scale Factor
allows you to specify a factor that scales the pseudo time step employed for the flow equations specified in the **Run Calculation** task page. A time scale factor other than 1.0 (default) allows the use of an equation specific time step in lieu of using a uniform global pseudo time step.

Non-Iterative Solver Controls
contain parameters that control the sub-iterations for the individual equations. See Time-Advancement Algorithm in the Theory Guide for details.

Max. Corrections
provide control over the maximum number of sub-iterations for each individual equation.

Correction Tolerance
defines the overall accuracy.

Residual Tolerance
controls the solution of the linear equations.

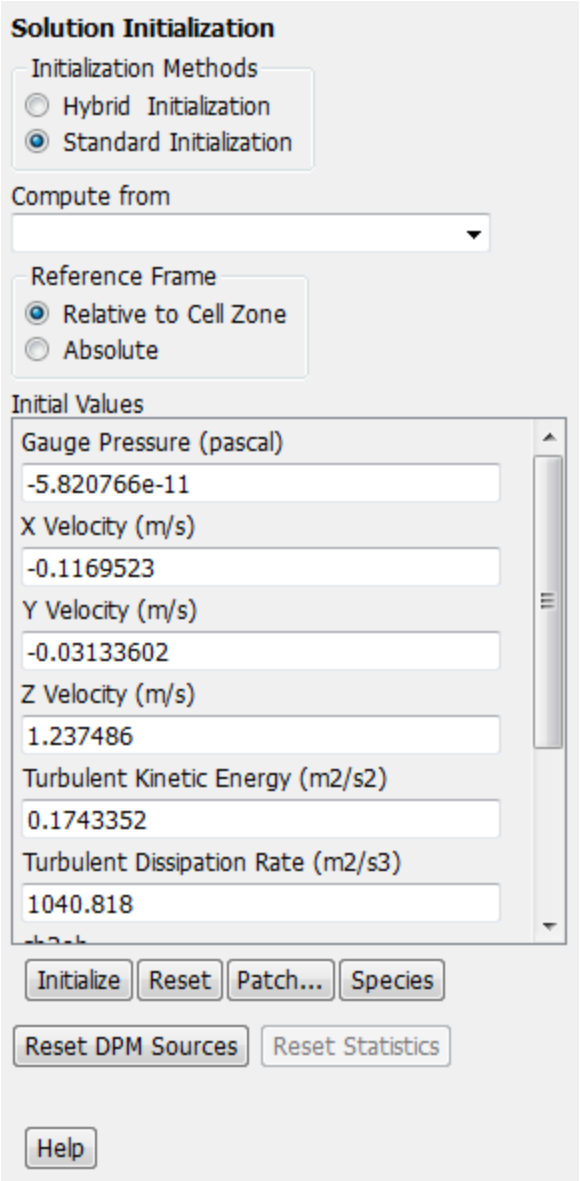
Default
sets the fields to their default values, as assigned by ANSYS Fluent. After execution, the **Default** button becomes the **Reset** button.

Reset
resets the fields to their most recently saved values (that is, the values before **Default** was selected). After execution, the **Reset** button becomes the **Default** button.

37.15. Solution Initialization Task Page

The **Solution Initialization** task page allows you to define values for flow variables and initialize the flow field to these values. See for details about using this dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Initialization Method

allows you to choose between **Hybrid Initialization** and **Standard Initialization**.

Hybrid Initialization

is a collection of boundary interpolation methods, where variables, such as temperature, turbulence, species fractions, volume fractions, and so on, are automatically patched based on domain averaged values or a particular interpolation recipe (see Hybrid Initialization (p. 1597)).

Standard Initialization

allows you to define values for flow variables and initialize the flow field to these values.

Compute from

is a drop-down list of zones; the default values for applicable variables will be computed from information contained in the zone that you select from this list. The computation will occur when you select the required zone, and the variable values will be displayed in **Initial Values**. You can also choose the **all-zones** item in this list to compute average values based on all zones.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reference Frame
indicates whether the initial velocities are absolute velocities (**Absolute**) or velocities relative to the motion of each cell zone (**Relative to Cell Zone**). This selection is necessary only if your problem involves moving reference frames or sliding meshes. If there is no zone motion, both options are equivalent.

Initial Values
displays the initial values of applicable variables. You can use **Compute from** to compute values from a particular zone, or you can enter values directly. For species transport cases with a single phase flow, only the boundary species that you have selected in the **Select Boundary Species** dialog box (see Figure 14.3: The Select Boundary Species Dialog Box (p. 966)) appear in the list. You can add or remove a species using the **Select Boundary Species** dialog box that can be accessed by clicking **Species** in the **Initialization** task page.

Initialize
initializes the entire flow field to the values listed.

Reset
resets the fields to their "saved" values.

Patch...
opens the Patch Dialog Box (p. 2522).

Species
opens the **Select Boundary Species** dialog box (see Figure 14.3: The Select Boundary Species Dialog Box (p. 966)), where you can select the boundary species to be displayed in the **Initial Values** list and in the **Patch**, **Hybrid Initialization**, and relevant boundary conditions dialog boxes. You can modify the selected boundary species list on the fly during the problem setup. This item appears for species transport cases with a single phase flow only.

Reset DPM Sources
allows you to reset the interphase sources/sinks of momentum, heat, and/or mass to zero. This item is available when you perform coupled discrete phase calculations. See Resetting the Interphase Exchange Terms (p. 1329) for details.

Reset Statistics
can be used to both initialize the flow statistics and reset the flow statistics after you have gathered some data for time statistics. This item is available when you perform unsteady calculations and have enabled the **Data Sampling for Time Statistics** option in the Run Calculation Task Page (p. 2537). See User Inputs for Time-Dependent Problems (p. 1609) for details.

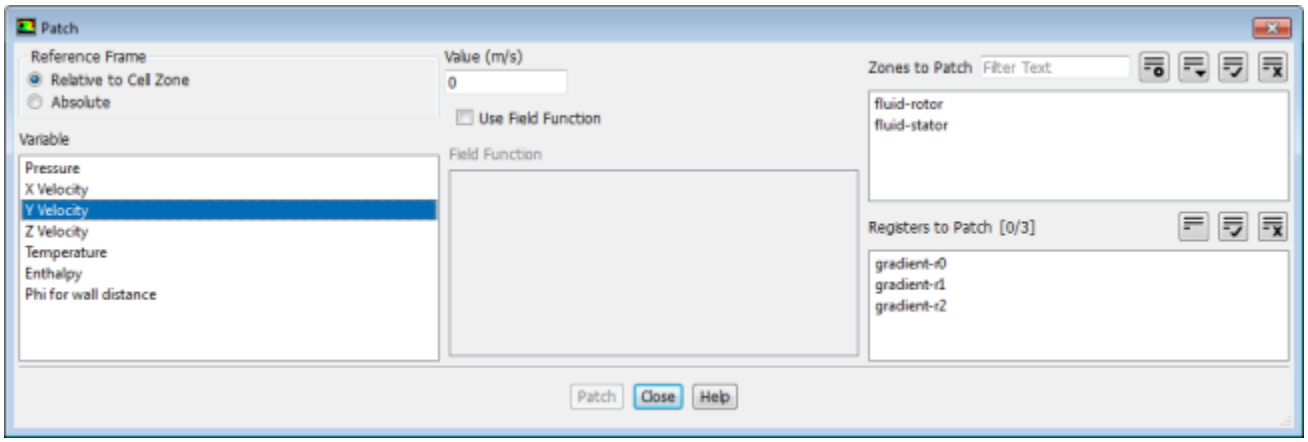
More Settings...
opens the Hybrid Initialization Dialog Box (p. 2524). This button is available only when **Hybrid Initialization** is the selected method.

For additional information, see the following sections:
37.15.1. Patch Dialog Box
37.15.2. Hybrid Initialization Dialog Box

37.15.1. Patch Dialog Box

The **Patch** dialog box allows you to patch different values of flow variables into different cells. See Patching Values in Selected Cells (p. 1593) for details about using this feature.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Reference Frame

indicates whether patched velocities are absolute velocities (**Absolute**) or velocities relative to the motion of each cell zone (**Relative to Cell Zone**). This selection is necessary only if your problem involves moving reference frames or sliding meshes. If there is no zone motion, both options are equivalent. Also, this selection affects only velocities, so the options will not be available unless you have selected a velocity in the **Variable** list.

Phase

contains a list of all of the phases in the problem that you have defined. This is available when the VOF, mixture, or Eulerian multiphase model is enabled.

Variable

contains a list of flow variables from which you can choose the variable to be patched. For species transport cases with a single phase flow, only the boundary species that you have selected in the **Select Boundary Species** dialog box (see Figure 14.3: The Select Boundary Species Dialog Box (p. 966)) appear in the list. You can add or remove a species using the **Select Boundary Species** dialog box that can be accessed by clicking **Species** in the **Initialization** task page.

Volume Fraction Patch Options

contains options that can be used when patching volume fraction for VOF and Eulerian multiphase with Multi-Fluid VOF simulations.

Patch Reconstructed Interface

applies a piecewise-linear reconstruction of the interface when patching a user-defined register and patches cells that intersect the interface with the actual volume fraction.

Volumetric Smoothing

enables smoothing of the volume fraction based on the volumetric average over the cell neighbors. After enabling volumetric smoothing, you can click Smooth to apply smoothing to the volume fraction field for all phases in the entire domain.

Smoothing Relaxation Factor

controls the degree of smoothing applied to the volume fraction field. This can be set between `0` (no smoothing) and `1` (maximum smoothing).

Value

sets a constant value to be patched for the selected **Variable**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use Field Function
enables the patching of a custom **Field Function**, rather than a constant **Value**, for the selected **Variable**. See Using Field Functions (p. 1595) for details.

Field Function
contains a list of defined custom field functions. If the **Use Field Function** option is enabled, you can patch one of these functions for the selected **Variable**. See Using Field Functions (p. 1595) for details.

Zones to Patch
contains a list of cell zones. The specified **Value** or **Field Function** for the selected **Variable** will be patched into the zones you select from this list.

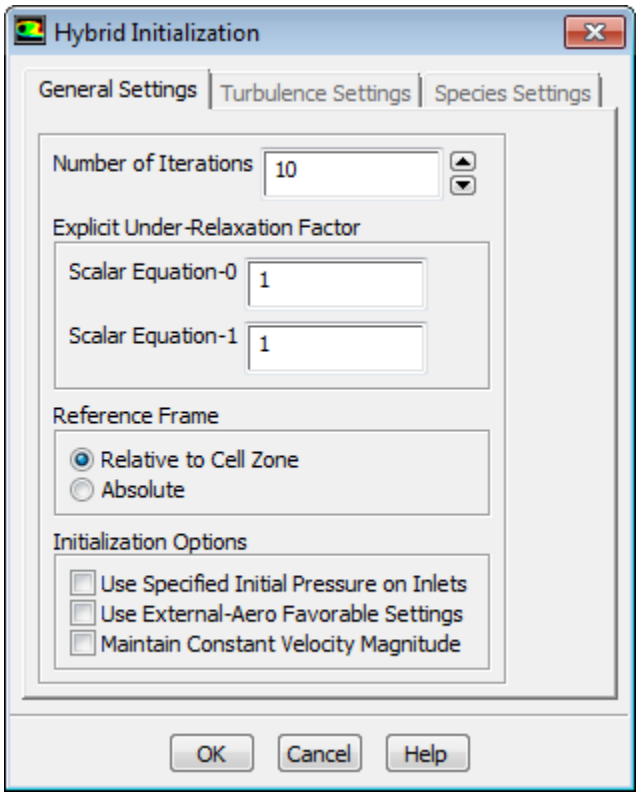
Registers to Patch
contains a list of cell registers that have been created using the adaption tools. You can patch a different value for a group of cells within a single cell zone by selecting a register containing a subset of the cells in the zone. See Using Registers (p. 1595) for details.

Patch
updates the flow-field data based on the inputs above.

Smooth
apply volumetric smoothing to the volume fraction field. This button only appears if **Volumetric Smoothing** is enabled.

37.15.2. Hybrid Initialization Dialog Box

The **Hybrid Initialization** dialog box contains a host of adjustable settings that control the **Hybrid Initialization** strategy.



Controls

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Settings

allow you to adjust such entries as the number of iterations and under-relaxation factors.

Number of Iterations

uses a default value of 10. This is the number of iterations that will be performed while solving the Laplace equations to initialize the velocity and pressure. If the initialized velocity and pressure fields are not to your liking, you may want to increase the number of iterations and re-initialize the solution.

Explicit Under-Relaxation Factor

uses a default value of 1. This value will be used while solving the Laplace equation to initialize the velocity and pressure. If the initialized velocity and pressure fields are not to your liking, you may want to adjust the under-relaxation factor and re-initialize the solution.

Reference Frame

indicates whether the initial velocities are absolute velocities (**Absolute**) or velocities relative to the motion of each cell zone (**Relative to Cell Zone**). This selection is necessary only if your problem involves moving reference frames or sliding meshes. If there is no zone motion, both options are equivalent.

Initialization Options

allows you to include any of the three initialization options.

Use Specified Initial Pressure on Inlets

if you want the specified pressure for **Supersonic/Initialization Gauge Pressure** at the inlet boundaries to be used for solving the Laplace equation for the pressure.

Use External-Aero Favorable Settings

if you want to have the velocity potential patched with a linear value to help accelerate convergence of **Scalar Equation–0** and to obtain a better guess of the velocity field for external-aero problems.

Maintain Constant Velocity Magnitude

if you want to use the flow direction obtained from solving the velocity potential (**Scalar Equation–0**), while maintaining a constant velocity magnitude throughout the computational domain.

Turbulence Settings

uses by default the domain averaged values for the turbulence parameters. If you want to use variable turbulence parameters you can deselect the **Average Turbulent Parameters** check box. When this option is disabled, then it calculates the turbulent parameters, such as kinetic energy, dissipation energy, and so on, using local flow parameters.

Species Settings

will by default initialize secondary species with zero mass or mole fractions. If you want to specify the appropriate value for the species, you must enable **Specify Species Parameters**. Note that only the boundary species that you have selected in he **Select Boundary Species** dialog box (see Figure 14.3: The Select Boundary Species Dialog Box (p. 966)) appear in the list. You can add or remove a species using the **Select Boundary Species** dialog box that can be accessed by clicking **Species** in the **Initialization** task page.

37.16. Calculation Activities Task Page

The **Calculation Activities** task page allows you to set up various tasks that you can perform during the calculation, such as saving files, exporting files, creating solution animations, and command execution.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Autosave Every
allows you to set the frequency of the autosave.

Edit...
opens the Autosave Dialog Box (p. 2527).

Automatic Export
displays a list of available export objects that will be executed during the calculations. This list and its associated buttons are only available for transient flow calculations.

Create
provides a drop-down list that contains options for creating export objects. Two options are available: the **Solution Data Export** option opens the Automatic Export Dialog Box (p. 2530); the **Particle History Data Export** option opens the Automatic Particle History Data Export Dialog Box (p. 2533).

Edit...
opens the appropriate dialog box for the selected item in the **Automatic Export** list.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Delete
removes the selected item from the **Automatic Export** list.

Execute Commands
lists available commands to be executed during the calculations.

Create/Edit...
opens the Execute Commands Dialog Box (p. 2535).

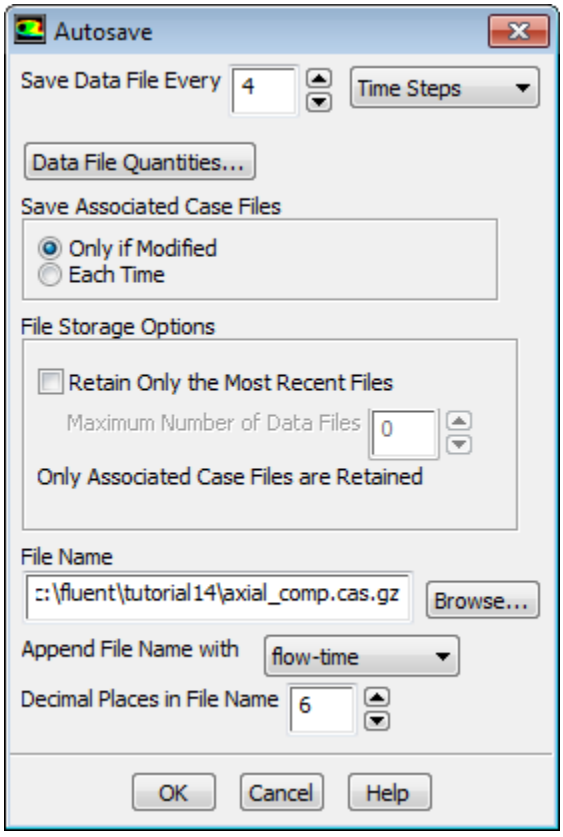
Automatically Initialize Solution and Modify Case
allows you to automatically have your solution initialized and your case file modified.

Edit...
opens the Automatic Solution Initialization and Case Modification Dialog Box (p. 2536). This button is only available when the **Automatically Initialize Solution and Modify Case** option is enabled.

For additional information, see the following sections:
37.16.1. Autosave Dialog Box
37.16.2. Data File Quantities Dialog Box
37.16.3. Automatic Export Dialog Box
37.16.4. Automatic Particle History Data Export Dialog Box
37.16.5. Execute Commands Dialog Box
37.16.6. Define Macro Dialog Box
37.16.7. Automatic Solution Initialization and Case Modification Dialog Box

37.16.1. Autosave Dialog Box

The **Autosave** dialog box allows you to specify automatic saving of case and data files at specified intervals during a calculation. See Automatic Saving of Case and Data Files (p. 48) for details.



Controls

Save Data File Every
specifies the frequency with which data files are saved. For steady flows you will specify the frequency in iterations, while for unsteady flows you will specify it in time steps (unless you are using the explicit time stepping formulation, in which case you will specify the frequency in iterations or if you are running an in-cylinder simulation you can specify the frequency in crank angles). The default value is set to zero, indicating that no automatic saving is performed.

When the Data File is saved, Save the Case
gives you the choice to save the case file only if it is modified or each time that the data file is saved.

If Modified During the Calculation or Manually
will result in ANSYS Fluent saving a case file whether you make a manual change, or if ANSYS Fluent makes a change to the code internally during the calculation.

Each Time
allows you to save the case file every time the data file is saved.

Retain Only the Most Recent Files
allows you to restrict the number of files saved by ANSYS Fluent if you have limited disk space. When this option is enabled, you can enter the appropriate value in the **Maximum Number of Data Files** field.

Only the associated case files are retained when using this option.

Important

When the **Retain Only the Most Recent Files** option is selected, the solution history currently in memory will be discarded and the solution history reset.

Data File Quantities
opens the Data File Quantities Dialog Box (p. 2529) where you can specify which quantities you want to automatically save to a data file for postprocessing.

Maximum Number of Data Files
specifies the maximum number of data files that can be saved at any instance. If you have constraints on the disk space, you can restrict the number of files to be saved using this field. After saving the specified number of files, ANSYS Fluent will overwrite the earliest existing file. The default value for this field is zero, which saves all the files.

File Name
specifies the root name for the files that are saved. The iteration or time-step number and an appropriate suffix (`.cas` or `.dat`) will be added to the specified root name. If the specified **File Name** ends in `.gz` or `.Z`, appropriate file compression will be performed. (See Reading and Writing Compressed Files (p. 41) for details about file compression.)

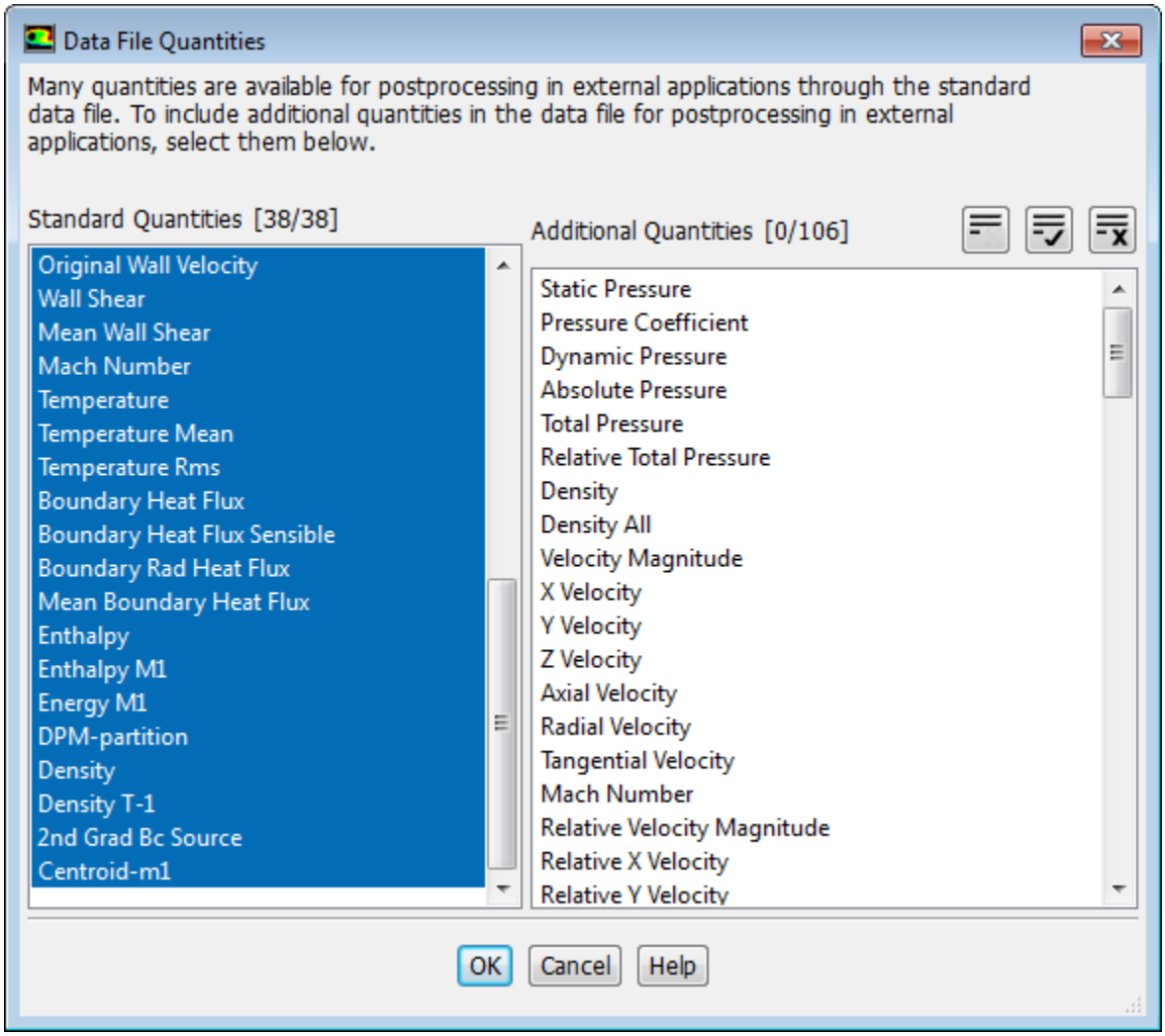
Append File Name with
allows you to select **flow-time**, **time-step**, or **crank-angle** to be appended to the file name. This option is available only for unsteady-state calculations (only for in-cylinder calculations in the case of **crank-angle**). The default selection is **flow-time**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Decimal Places in File Name
allows you to specify the number of decimal digits in the file name. This option is available only when **flow-time** or **crank-angle** is selected in the **Append File Name with** drop-down list. The default value for this field is set to 6.

37.16.2. Data File Quantities Dialog Box

The **Data File Quantities** dialog box allows you to specify various quantities for postprocessing. See Setting Data File Quantities (p. 107) for details.



Controls

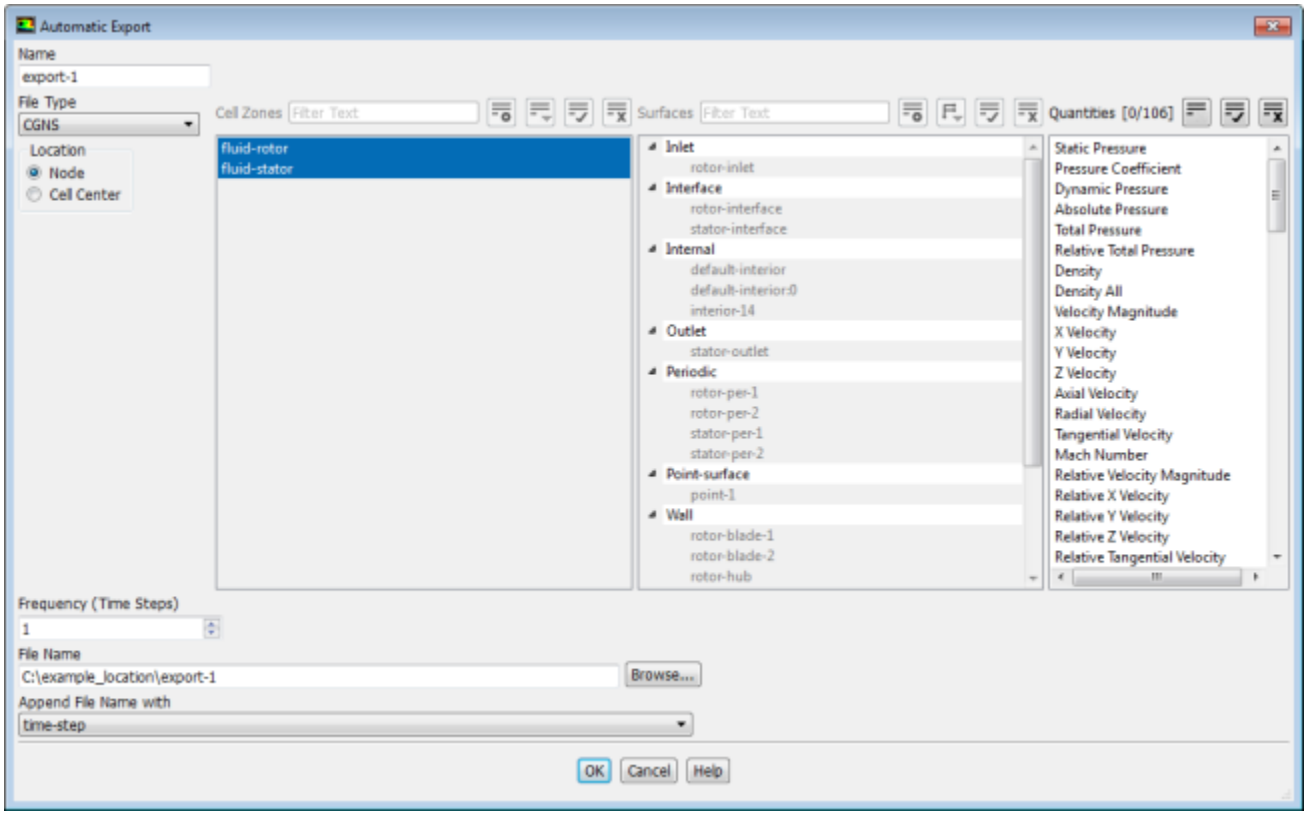
Standard Quantities
contains a listing of standard postprocessing quantities (for example, density, Mach number, temperature, and so on).

Additional Quantities
contains a listing of additional postprocessing quantities that are derived from the standard quantities (for example, standard pressure, velocity magnitude, and so on).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.16.3. Automatic Export Dialog Box

The **Automatic Export** dialog box allows you to create an automatic export definition for solution data. See Creating Automatic Export Definitions for Solution Data (p. 87) for details.



Controls

Name
specifies the name of the export definition.

File Type
contains a drop-down list of file types, which control the output file format that will be written.

ABAQUS
allows you to specify the surface(s) and optional loads, based on the kind of finite element analysis selected, to be exported to an ABAQUS file (extension `.inp`).

ASCII
allows you to specify the surface(s), scalars, location from which the values of scalar functions are to be taken, and the delimiter separating the fields, to be exported to an ASCII file.

AVS
allows you to specify the scalars you want to write to be exported to an AVS file.

CFD-Post Compatible
allows you to specify the scalars you want to write, the cell zones from which the values of scalar functions are to be taken, the file format, and whether case files are written with every `.cdat` file, as part of the exporting of files that are compatible with the CFD-Post application (that is, `.cdat` and `.cst` files).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CGNS
allows you to specify the scalars you want to write and the location from which the values of scalar functions are to be taken, to be exported to a CGNS file (extension `.cgns`).

Data Explorer
allows you to specify the surface(s) and the scalars you want to write to be exported to a Data Explorer file (extension `.dx`).

EnSight Case Gold
allows you to specify the scalars you want to write, the cell zones, interior zone surfaces, and location in the cell from which the values of scalar functions are to be taken, and the file format, to be exported to an EnSight file (extension `.geo`, `.vel`, `.scl1`, or `.encas`).

FAST
allows you to specify the scalars you want to write, to be exported as a grid file (Plot3D format), a velocity file, and a scalar file. This option is available only for a triangular or tetrahedral mesh.

FAST Solution
allows you to export a single file containing density, velocity, and total energy data. This option is available only for a triangular or tetrahedral mesh.

Fieldview Unstructured
allows you to specify the scalars you want to write and the cell zones from which the values of scalar functions are to be taken, to be exported to a FIELDVIEW binary file (extension `.fvuns`) and a regions file (extension `.fvuns.fvreg`).

I-deas Universal
allows you to specify the surface(s), scalars, and optional loads, based on the kind of finite element analysis selected, to be exported to an I-deas Universal file.

Mechanical APDL Input
allows you to specify the surface(s) and optional loads, based on the kind of finite element analysis selected, to be exported to a Mechanical APDL Input file (extension `.cdb`).

NASTRAN
allows you to specify the surface(s), scalars, and optional loads, based on the kind of finite element analysis selected, to be exported to a NASTRAN file (extension `.bdf`).

PATRAN
allows you to specify the surface(s), scalars, and optional loads, based on the kind of finite element analysis selected, to be exported to a PATRAN neutral file (extension `.out`).

RadTherm
allows you to specify the surface(s) for which you want to write data and the method of writing the heat transfer coefficient, to be exported to a PATRAN neutral file (extension `.neu`). This option is available only when the **Energy Equation** is enabled.

Tecplot
allows you to specify the surface(s) and the scalars you want to write, to be exported to a Tecplot file.

Cell Zones
specifies the cell zones for which data is to be written for a CFD-Post compatible, EnSight, or FIELDVIEW file.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surfaces
specifies the surfaces for which data is to be written for an ABAQUS, ASCII, Data Explorer, I-deas Universal, Mechanical APDL Input, NASTRAN, PATRAN, RadTherm, or Tecplot file.

Quantities
specifies valid quantities for output. The attributes of the list are modified based on the active file type. The list may be a single-selection or a multiple-selection list or it may be disabled, depending on the selected **File Type**.

Analysis (for ABAQUS, I-deas Universal, Mechanical APDL Input, NASTRAN, and PATRAN formats)
specifies the finite element analysis intended.

Structural
specifies structural analysis and allows you to select the **Structural Loads** to be written.

Thermal
specifies thermal analysis and allows you to select the **Thermal Loads** to be written.

Structural Loads
contains optional structural loads that can be written to ABAQUS, I-deas Universal, Mechanical APDL Input, NASTRAN, and PATRAN files. This option is available only when **Structural** analysis is selected.

Force
enables force to be written as a load for a structural analysis.

Pressure
enables pressure to be written as a load for a structural analysis.

Temperature
enables temperature to be written as a load for a structural analysis. This option is available only when the **Energy Equation** is enabled.

Thermal Loads
contains optional thermal loads that can be written to ABAQUS, I-deas Universal, Mechanical APDL Input, NASTRAN, and PATRAN files. This option is available only when **Thermal** analysis is selected.

Temperature
enables temperature to be written as a load for a thermal analysis.

Heat Flux
enables heat flux to be written as a load for a thermal analysis.

Heat Trans Coeff
enables heat transfer coefficient to be written as a load for a thermal analysis.

Location (for ASCII, CGNS, and EnSight Case Gold formats)
specifies the location from which the values of scalar functions are to be taken.

Node
specifies that data values at the node points are to be exported.

Cell Center
specifies that data values from the cell centers are to be exported.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Format (for CFD-Post Compatible and EnSight Case Gold)
specifies the file format.

Binary
specifies the file format as binary.

ASCII
specifies the file format as ASCII.

Heat Transfer Coefficient (for RadTherm format only)
specifies the basis for the heat transfer coefficient exported.

Flux Based
specifies the flux based method for writing the heat transfer coefficient.

Wall Function
specifies the wall function based method for writing the heat transfer coefficient.

Write Case File Every Time (for CFD-Post Compatible)
specifies whether a case file is written with every `.cdat` file, or if case files are written based on the settings specified by the `file/transient-export/settings/cfd-post-compatible` text command.

Frequency (Time Steps)
specifies the frequency for appending the data during the solution process.

File Name
specifies the root name for the files to be saved.

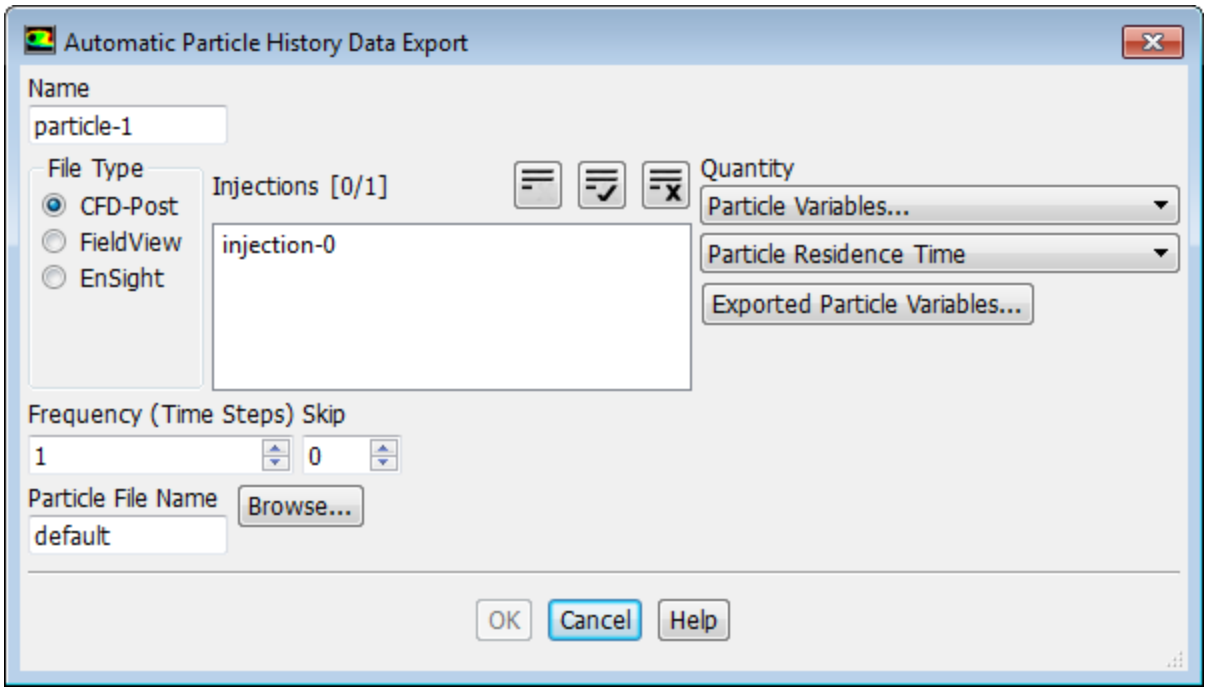
Append File Name with
allows you to select **flow-time** or **time-step** to be appended to the file name.

Decimal Places in File Name
allows you to specify the number of decimal digits in the file name. This option is available only when **flow-time** is selected in the **Append File Name with** drop-down list. The default value for this field is set to 6.

37.16.4. Automatic Particle History Data Export Dialog Box

The **Automatic Particle History Data Export** dialog box allows you to create an automatic particle history export definition for solution data. See Creating Automatic Export Definitions for Transient Particle History Data (p. 89) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name
specifies the name of the particle history export definition.

File Type
specifies the type of the file you want to write.

CFD-Post
allows you to write the file in CFD-Post particle tracks format, which can be read in **CFD-Post**.

FieldView
allows you to write the file in **FIELDVIEW** format, which can be read in **FIELDVIEW**.

EnSight
allows you to write the file in **EnSight** format.

Injections
allows you to select the required injection from the list of predefined injections.

Quantity
contains the list of variables for which you can export the particle data.

Skip
allows you to "thin" or "sample" the number of particles that are exported.

Frequency (Time Steps) or (DPM Iterations)
specifies the frequency of particle time steps that are used for saving the export file.

Separate Files for Each Time Step
allows you to have separate exported data files for each time step. Available only when **EnSight** is selected as the **File Type**.

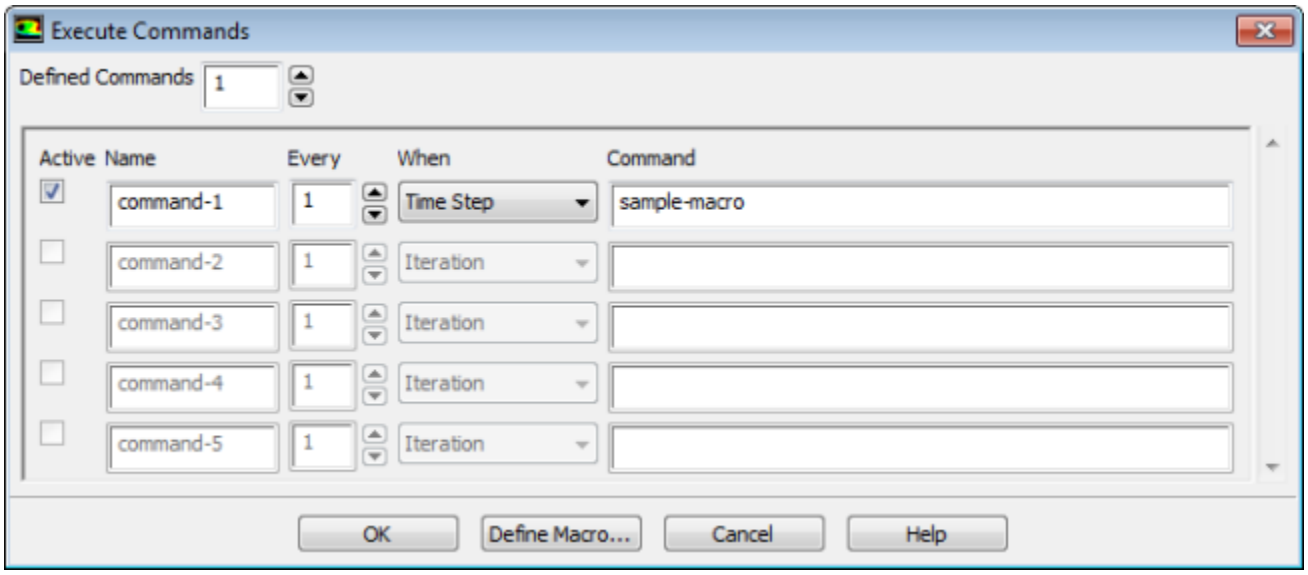
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Particle File Name
allows you to specify the file name/directory for the exported data, using the **Browse...** button.

Ensight Encas File Name
is the file name you will specify if you selected **EnSight** under **File Type**. Use the **Browse...** button to select the `.encas` file that was created when you exported the file with the **File/Export...** ribbon tab option.

37.16.5. Execute Commands Dialog Box

The **Execute Commands** dialog box allows you to define commands to be executed during the calculation. See Executing Commands During the Calculation (p. 1638) for details about using this feature.



Controls

Defined Commands
sets the total number of monitor commands to be defined.

Active
activates/deactivates the execution of each command.

Name
specifies a name for each command.

Every, When
indicate how often the command is to be executed. You can enter the interval under **Every** and select **Iteration** or **Time Step** under **When**. (**Time Step** is a valid choice only if you are calculating unsteady flow.)

Command
specifies the command to be executed. You can enter text commands or the name of a command macro that you have defined in the Define Macro Dialog Box (p. 2536).

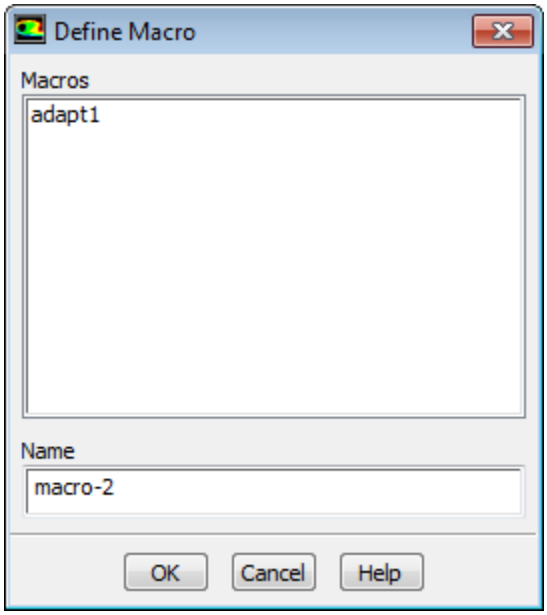
Define Macro...
opens the Define Macro Dialog Box (p. 2536), in which you can define command macros.

End Macro
ends the definition of a macro. (This button will replace the **Define Macro...** button when you click **OK** in the **Define Macro** dialog box.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.16.6. Define Macro Dialog Box

The **Define Macro** dialog box allows you to define macros for automatic execution by the command monitor, or for interactive use by you. See Defining Macros (p. 1640) for details.



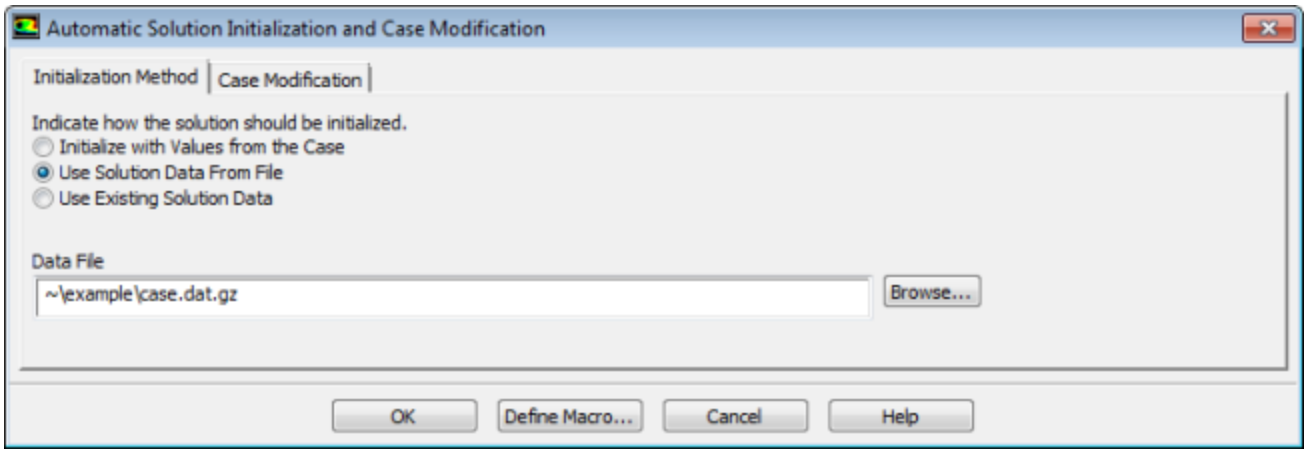
Controls

Macros
contains a selectable list of the currently-defined macros.

Name
specifies a name for the command macro.

37.16.7. Automatic Solution Initialization and Case Modification Dialog Box

The **Automatic Solution Initialization and Case Modification** dialog box allows you to specify the solution initialization method and to modify the case. See Automatic Initialization of the Solution and Case Modification (p. 1642) for details.



Controls

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Initialization Method
tab contains several choices for initializing the solution.

Initialize with Values from the Case
uses the values set in the **Solution Initialization** task page.

Use Solution Data From File
requires you to read in a data file containing the desired initialization for the case.

Use Solution Data From Previous Parametric Run with Workbench
(transient cases only) requires you to select one of two methods to initialize the first run. You can **Initialize with Values from the Case**, which uses the values set in the **Solution Initialization** task page. Otherwise, you can **Use Solution Data from File**, which requires you to read in a data file containing the desired initialization for the case.

Use Existing Solution Data
is analogous to changing the values in a case and continuing the calculation. However, the iteration counter will be reset to 0 so that the modifications can be applied. Use this method when no solution data exists, similar to the first run.

Case Modification
allows you to indicate how long you would like to run with the original settings, then make any modifications to the case settings.

Defined Modifications
indicates the number of modifications for the case file.

Active
allows you to enable or disable a defined case modification.

Name
represents the name of the case modification.

Commands
represents an area where you can enter text commands.

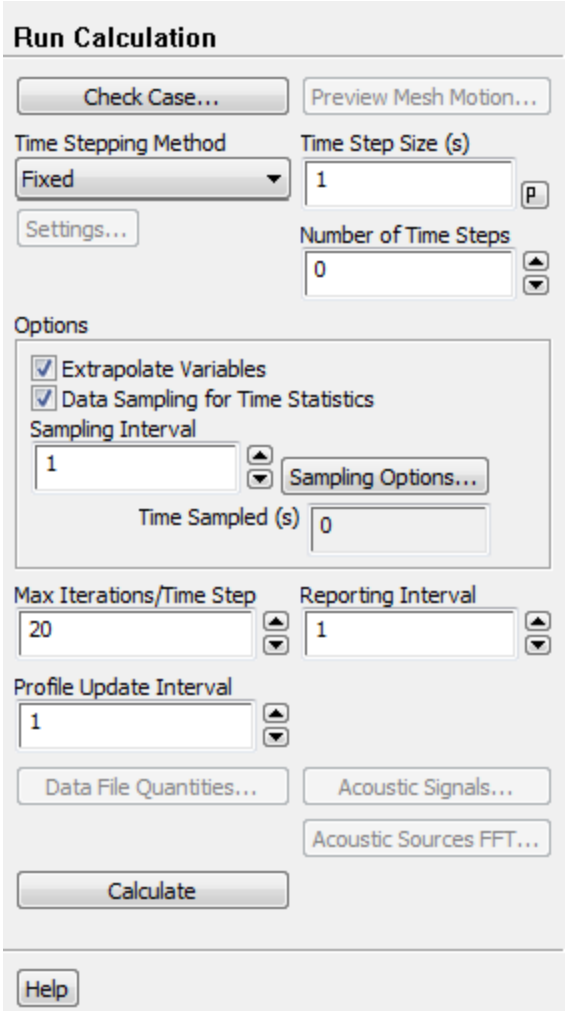
Number of Iterations/Time Steps
represents the number of iterations or time steps that you want to run defined case modification commands.

Define Macro...
opens the Define Macro Dialog Box (p. 2536), in which you can define command macros.

37.17. Run Calculation Task Page

The **Run Calculation** task page allows you to start the solver iterations. See Performing Steady-State Calculations (p. 1600) and Performing Time-Dependent Calculations (p. 1608) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Check Case...
opens the Case Check Dialog Box (p. 2543).

Preview Mesh Motion...
opens the Mesh Motion Dialog Box (p. 2500).

Pseudo Transient Options
is available for fluid and solid zones and will only appear for steady-state cases, where the pressure-based coupled solver is used. Note that the **Pseudo Transient** option must be enabled in the Solution Methods Task Page (p. 2504). For details about the available options, see Performing Pseudo Transient Calculations (p. 1601).

Number of Iterations
(for steady flow calculations) sets the number of iterations to be performed. (For unsteady calculations using the explicit unsteady formulation, this will specify the number of time steps, since each iteration will be a time step.)

Time Stepping Method
(for transient flow calculations) contains options for how the time step is determined.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fixed
selects a fixed time step, equal to the specified **Time Step Size**.

Adaptive
selects a time step that is initially equal to the specified **Time Step Size**, but gets modified by ANSYS Fluent as the calculation proceeds. See Adaptive Time Stepping (p. 1618) for details.

Variable
enables variable time stepping method. The inputs are same as for the adaptive time stepping method, with the exception of specifying a global Courant number. **Variable** is selectable when the VOF multiphase model is enabled. See Variable Time Stepping (p. 1621) for details.

Settings...
opens either the Adaptive Time Step Settings Dialog Box (p. 2544) when **Adaptive** is selected as the **Time Stepping Method**, or the Variable Time Step Settings Dialog Box (p. 2545) when **Variable** is selected as the **Time Stepping Method**.

Time Step Size(s)
(for transient flow calculations) sets the magnitude of the (physical) time step Δt.

Time Step Size for Acoustic Data Export
determines the highest frequency that the acoustic analysis reproduces. This is available when the FW-H acoustics model is enabled.

Number of Time Steps
(for transient flow calculations) sets the number of time steps to be performed.

Options
contains options related to unsteady calculations.

Extrapolate Variables
instructs ANSYS Fluent to predict the solution variable values for the next time step and then input that predicted value as an initial guess for the inner iterations of the current time step.

Data Sampling for Time Statistics
enables the sampling of data during an unsteady calculation. See User Inputs for Time-Dependent Problems (p. 1609) and Postprocessing for Time-Dependent Problems (p. 1622) for details.

Sampling Interval
allows you to specify the frequency of the **Data Sampling for Time Statistics**.

Sampling Options...
opens the Sampling Options Dialog Box (p. 2546) for the **Data Sampling for Time Statistics**.

Time sampled
displays the time period over which data has been sampled for the postprocessing of the mean and RMS values.

Solid Time Step
enables a different time step to be specified for solid zones, either user specified or automatically calculated. See Specifying a Solid Time step (p. 829) for details.

Max Iterations/Time Step
for unsteady flow calculations using an implicit unsteady formulation) sets the maximum number of iterations to be performed per time step. If the convergence criteria are met before this number of iterations

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

is performed, the solution will advance to the next time step. See User Inputs for Time-Dependent Problems (p. 1609) for details.

Reporting Interval
sets the number of iterations that will pass before convergence monitors will be printed and plotted. The default is 1 (that is, reports will be updated after each iteration).

Postprocess Pollutants
results in the automatic postprocessing of pollutants during a transient simulation. This option is available when one of the pollution models is enabled.

Max Post Iterations/Time Step
sets the maximum number of postprocessing iterations to be performed per time step. This field is available when the **Postprocess Pollutants** option is enabled.

Profile Update Interval
sets the number of iterations that will pass before user-defined functions for boundary profiles will be updated.

Solution Steering
allows you to set parameters that will help ANSYS Fluent guide the calculations to a converged solution (available only for steady-state flows in the density-based implicit solver). See Solution Steering (p. 1675) for more information. When enabled, the following options are available:

Flow Type
allows you to select the flow type that best describes the flow in the solution domain. Five choices are available: **incompressible**, **subsonic**, **transonic**, **supersonic**, and **hypersonic**.

Use FMG Initialization
allows for full multigrid initialization.

First to Higher Order Blending
allows you to reduce the desired solution accuracy by selecting a blending factor less than 100%. The default setting is 100%. See First-to-Higher Order Blending in the Theory Guide for more information. The blending factor will be grayed out if **Second Order Upwind** discretization for the **Flow** equations is not selected in the **Solution Methods** task page. The solution accuracy may be reduced (typical values are 75% or 50%) if it is not possible to obtain a converged solution with the maximum second-order accuracy (blending = 100%).

More Settings...
opens the Solution Steering Dialog Box (p. 2541).

Courant Number
is a non-adjustable field displaying the current CFL number, which allows you to view it during the calculation.

Data File Quantities...
opens the Data File Quantities Dialog Box (p. 2529).

Acoustic Signals...
opens the Acoustic Signals Dialog Box (p. 2547). This button is only available when the acoustics model is enabled.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Acoustic Sources FFT...

opens the Acoustic Sources FFT Dialog Box (p. 2549). This button is only available for three-dimensional transient simulations, when the **Export Acoustic Source Data in ASD Format** option is enabled in the **Acoustics Model** dialog box.

Calculate

starts the calculations. While the calculation is in progress, a **Working** dialog box will appear. Clicking the **Cancel** button or typing **Ctrl**+**c** in the console window will interrupt the calculation at the earliest safe stopping point after the current iteration (in steady-state simulations) or the current time step (in transient simulations). The stopping point is chosen so that the field variables and solver state are updated and internally consistent. If you are running a transient simulation and want to stop the simulation immediately (before the completion of the current time step), you can type **Ctrl**+**c** a second time.

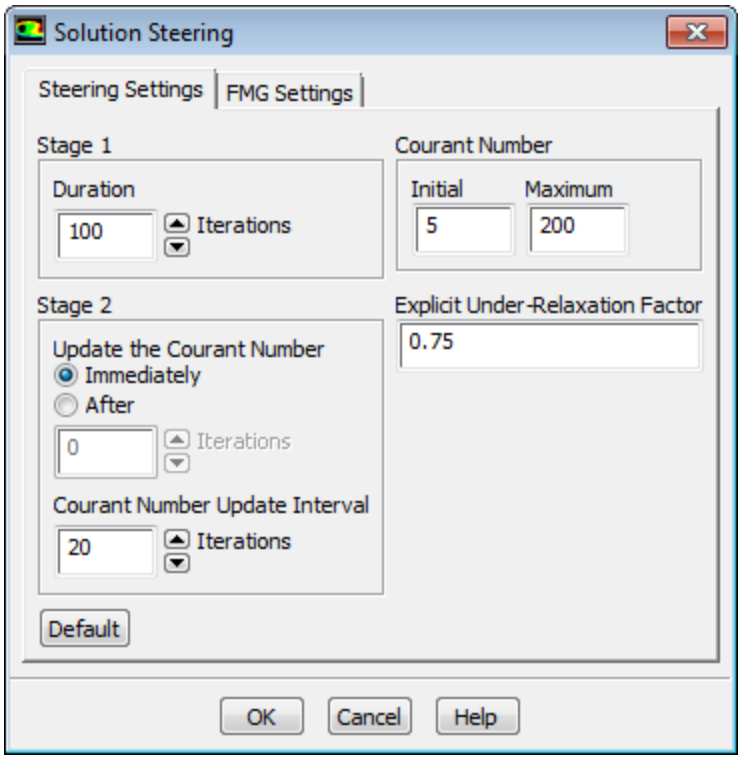
For additional information, see the following sections:

37.17.1. Solution Steering Dialog Box
37.17.2. Case Check Dialog Box
37.17.3. Adaptive Time Step Settings Dialog Box
37.17.4. Variable Time Step Settings Dialog Box
37.17.5. Sampling Options Dialog Box
37.17.6. Acoustic Signals Dialog Box
37.17.7. Acoustic Sources FFT Dialog Box

37.17.1. Solution Steering Dialog Box

The **Solution Steering** dialog box is used to set the parameters that control the solution steering strategy. Solution steering will typically perform full multigrid (FMG) initialization followed by two iterative stages (Stage 1 and Stage 2). The purpose of Stage 1 is to navigate the solution from the difficult initial phase of the solution toward convergence by insuring maximum stability. During this stage, the solution is advanced gradually from 1st-order accuracy to maximum accuracy (user specified and typically 2nd-order) at a constant low CFL value. In Stage 2, the solution is driven hard towards convergence by regular adjustments of the CFL value to insure fast convergence as well as to prevent possible divergence. In Stage 2, the residual history is monitored and analyzed through regular intervals to determine if an increase or decrease in CFL value is needed to obtain fast convergence or to prevent divergence. See Solution Steering (p. 1675) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Steering Settings
allows you to modify the steering parameters used in Stages 1 and 2.

Stage 1
allows you to set parameters relating to stage 1.

Duration
is the number of iterations in stage 1. The CFL number used during these iterations is set in the **Initial** field, in the **Courant Number** group box.

Stage 2
allows you to set parameters relating to stage 2.

Update the Courant Number
allows you to update the Courant number either **Immediately**, or **After** a specified number of iterations.

Courant Number Update Interval
defines the frequency at which the Courant number is updated.

Courant Number
allows you to set the starting (**Initial**) and maximum allowed (**Maximum**) Courant number values. The solution steering algorithm will not allow the solver to exceed the maximum Courant number, but will allow the solver to use a Courant number less than the initial Courant number if divergence in the solution has occurred.

Explicit Under-Relaxation Factor
allows the solution to be under-relaxed to improve convergence.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Default
resets any changes made to the parameters to their original default values.

FMG Settings
allows you to set FMG parameters. For more information about FMG initialization, refer to Full Multigrid (FMG) Initialization (p. 1595).

Number of Multigrid Levels
allows you to set the number of multigrid levels.

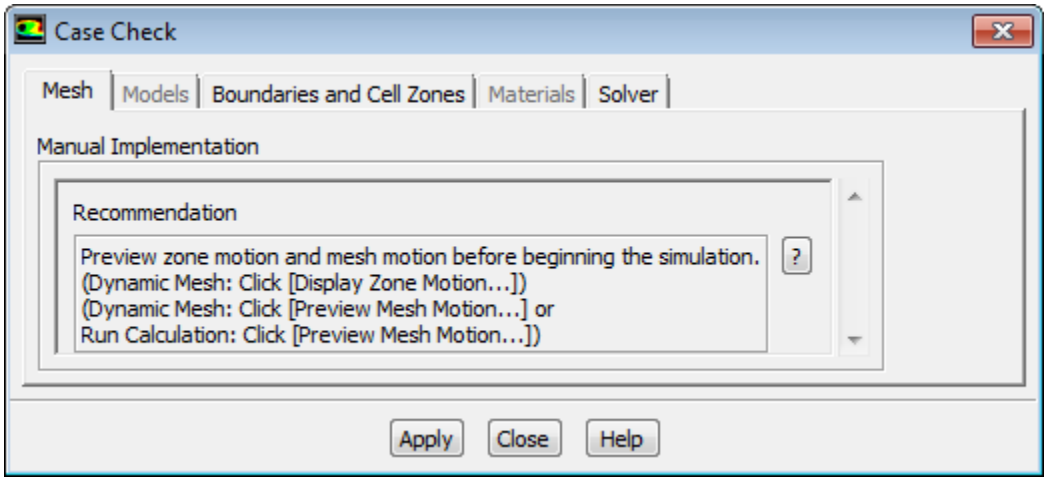
Number of Cycles
allows you to set the number of cycles for a selected level.

FMG Courant Number
allows you to set the FMG Courant number.

Default
resets any changes to the original default values.

37.17.2. Case Check Dialog Box

This function provides you with guidance and best practices when choosing case parameters and models. Your case will be checked for compliance in the mesh, models, boundary and cell zone conditions, material properties, and solver categories. Established rules are available for each category, with recommended changes to your current settings. Information about each of the recommendations is available in Checking Your Case Setup (p. 1654).



Controls

Mesh
displays recommendations, if any, relating the to the mesh used in the case. See Checking the Mesh (p. 1657) for details.

Models
displays recommendations, if any, relating the to the models used in the case. See Checking Model Selections (p. 1659) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

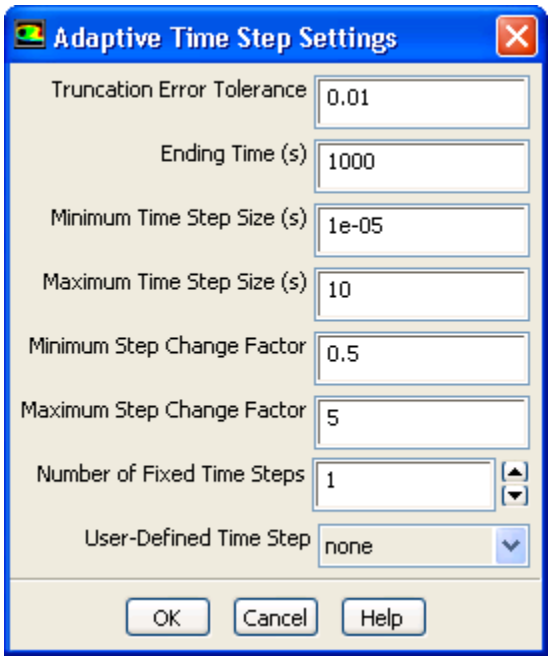
Boundaries and Cell Zones
displays recommendations, if any, relating the to the cell zones or boundaries defined in the case. See Checking Boundary and Cell Zone Conditions (p. 1661) for details.

Materials
displays recommendations, if any, relating the to the materials defined in the case. See Checking Material Properties (p. 1664) for details.

Solver
displays recommendations, if any, relating the to the solver settings used in the case. See Checking the Solver Settings (p. 1665) for details.

37.17.3. Adaptive Time Step Settings Dialog Box

The **Adaptive Time Step Settings** dialog box is used to set the parameters that control the adaptive time stepping. See Adaptive Time Stepping (p. 1618) for details.



Controls

Truncation Error Tolerance
specifies the threshold value to which the computed truncation error is compared. Increasing this value will lead to an increase in the size of the time step and a reduction in the accuracy of the solution. Decreasing it will lead to a reduction in the size of the time step and an increase in the solution accuracy, although the calculation will require more computational time. For most cases, the default value of 0.01 is acceptable.

Ending Time
specifies an ending time for the calculation. Since the ending time cannot be determined by multiplying the number of time steps by a fixed time step size, you need to specify it explicitly.

Minimum/Maximum Time Step Size
specify the upper and lower limits for the size of the time step. If the time step becomes very small, the computational expense may be too high; if the time step becomes very large, the solution accuracy may not be acceptable to you. You can set the limits that are appropriate for your simulation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Minimum/Maximum Step Change Factor

limit the degree to which the time step size can change at each time step. Limiting the change results in a smoother calculation of the time step size, especially when high-frequency noise is present in the solution. If the time step change factor, f, is computed as the ratio between the specified truncation error tolerance and the computed truncation error, the size of time step Δt_n is computed as follows:

- If $1 < f < f_{max}$, Δt_n is increased to meet the required tolerance.
- If $1 < f_{max} < f$, Δt_n is increased, but its maximum possible value is $f_{max} \Delta t_{n-1}$.
- If $f_{min} < f < 1$, Δt_n is unchanged.
- If $f < f_{min} < 1$, Δt_n is decreased.

Number of Fixed Time Steps

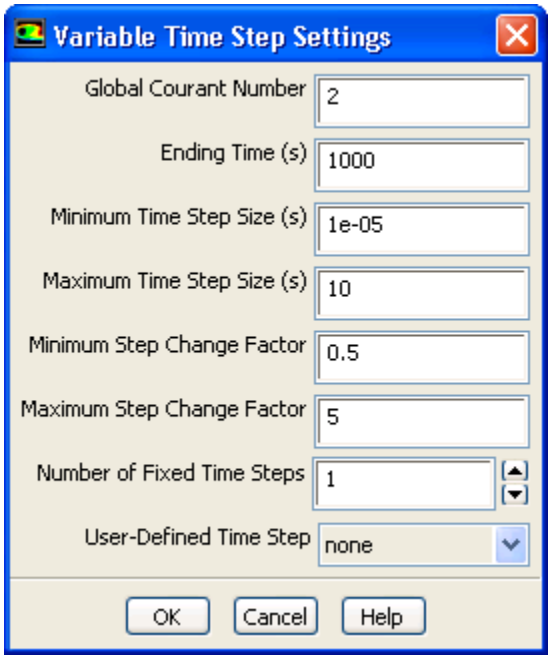
specifies the number of fixed-size time steps that should be performed before the size of the time step starts to change. The size of the fixed time step is the value specified for **Time Step Size** under **Time**.

User-Defined Time Step

contains a drop-down list of available user-defined functions.

37.17.4. Variable Time Step Settings Dialog Box

The **Variable Time Step Settings** dialog box contains parameters related to variable time stepping. See Variable Time Stepping (p. 1621) for details.



Controls

Global Courant Number

allows you to specify the Courant number. The default value for the **Global Courant number** is 2.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ending Time
specifies an ending time for the calculation. Since the ending time cannot be determined by multiplying the number of time steps by a fixed time step size, you need to specify it explicitly.

Minimum/Maximum Time Step Size
specify the upper and lower limits for the size of the time step. If the time step becomes very small, the computational expense may be too high; if the time step becomes very large, the solution accuracy may not be acceptable to you. You can set the limits that are appropriate for your simulation.

Minimum/Maximum Step Change Factor
limit the degree to which the time step size can change at each time step. Limiting the change results in a smoother calculation of the time step size, especially when high-frequency noise is present in the solution. If the time step change factor, f, is computed as the ratio between the specified truncation error tolerance and the computed truncation error, the size of time step Δt_n is computed as follows:

- If $1 < f < f_{max}$, Δt_n is increased to meet the required tolerance.
- If $1 < f_{max} < f$, Δt_n is increased, but its maximum possible value is $f_{max} \Delta t_{n-1}$.
- If $f_{min} < f < 1$, Δt_n is unchanged.
- If $f < f_{min} < 1$, Δt_n is decreased.

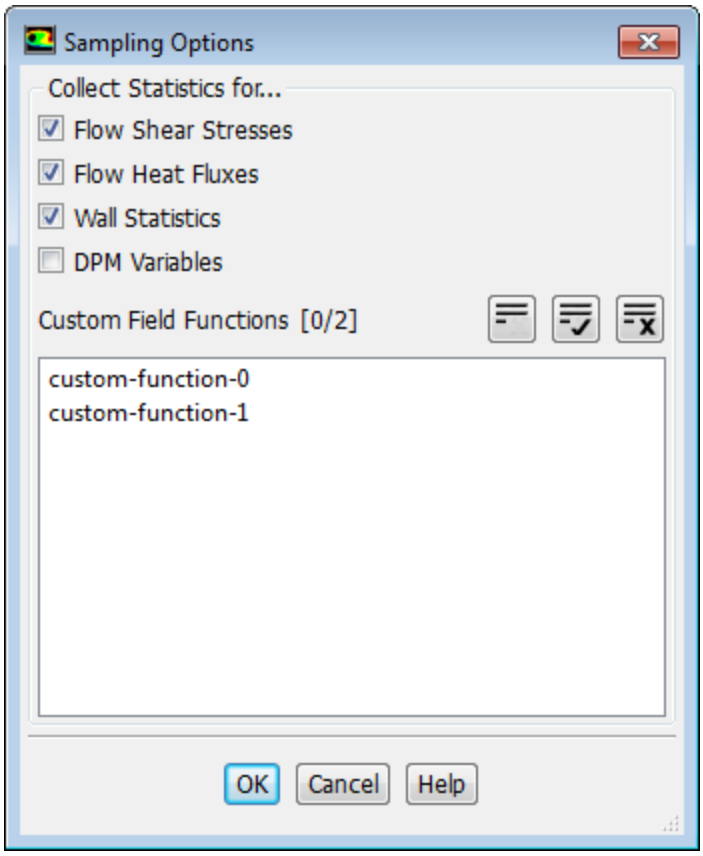
Number of Fixed Time Steps
specifies the number of fixed-size time steps that should be performed before the size of the time step starts to change. The size of the fixed time step is the value specified for **Time Step Size** under **Time**.

User-Defined Time Step
contains a drop-down list of available user-defined functions.

37.17.5. Sampling Options Dialog Box

The **Sampling Options** dialog box allows you to specify a collection of statistics for: **Flow Shear Stresses**, **Flow Heat Fluxes**, **Wall Statistics**, **DPM Variables**, and **Custom Field Functions**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Collect Statistics for...
contains options for the variables you want to be able to postprocess.

Flow Shear Stresses
allows you to enable or disable the flow shear stress statistics for postprocessing.

Flow Heat Fluxes
allows you to enable or disable the flow heat fluxes statistics for postprocessing.

Wall Statistics
allows you to enable or disable the wall statistics for postprocessing.

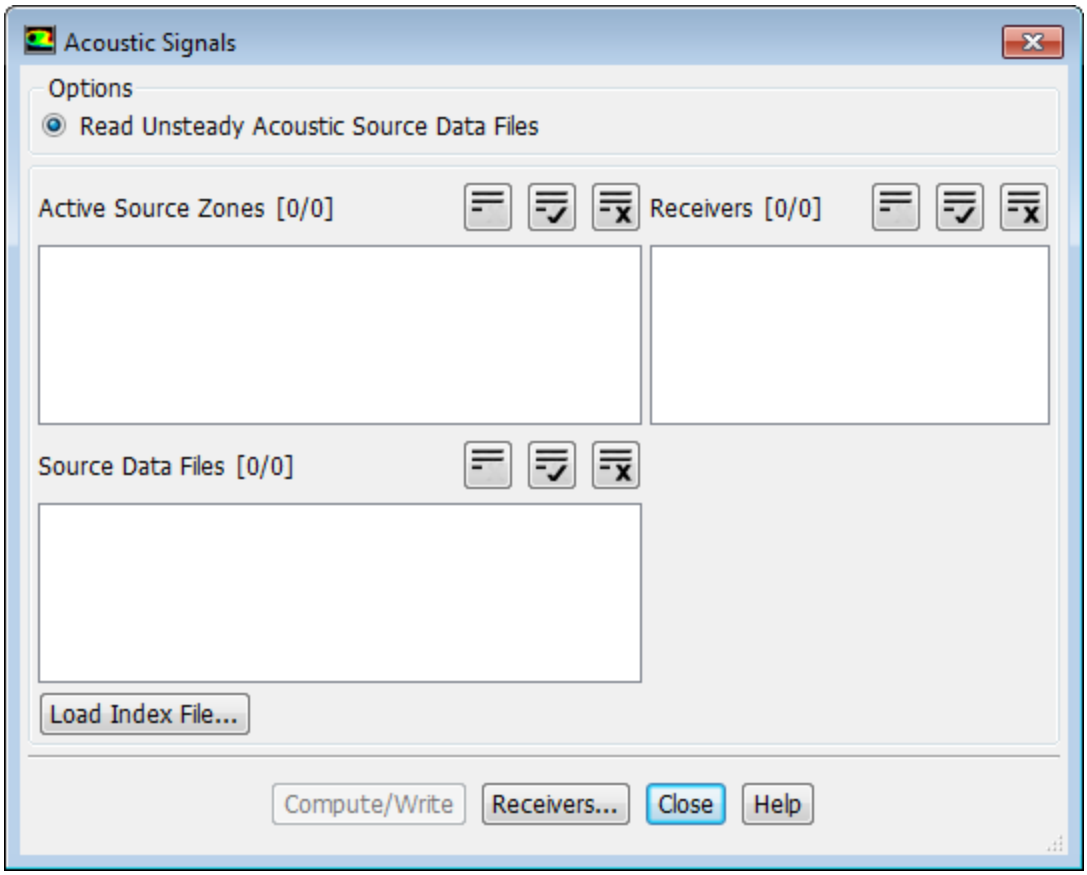
DPM Variables
allows you to enable or disable DPM variable statistics for postprocessing.

Custom Field Functions
allows you to select the previously defined custom field functions you want to be able to postprocess.

37.17.6. Acoustic Signals Dialog Box

The **Acoustic Signals** dialog box is used to compute and save the sound pressure signals. See Postprocessing the FW-H Acoustics Model Data (p. 1219) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options
contains the options available for acoustic signal postprocessing.

Write Acoustic Signals
enables the parameters needed to write the sound pressure data to files.

Read Unsteady Acoustic Source Data Files
enables the parameters needed to compute the sound pressure signals using the source data saved to files.

Active Source Zones
contains source zones you want to include to compute sound. See Specifying Source Surfaces (p. 1214) for details.

Receivers
contains all the receivers for which you can compute sound.

Source Data Files
contains all the source data files that you can use to compute sound.

Load Index File...
opens the **Select File** dialog box, in which you can select the index file for your computation.

Write
writes the sound pressure data. This button will appear only when **Write Acoustic Signals** is selected under **Options**.

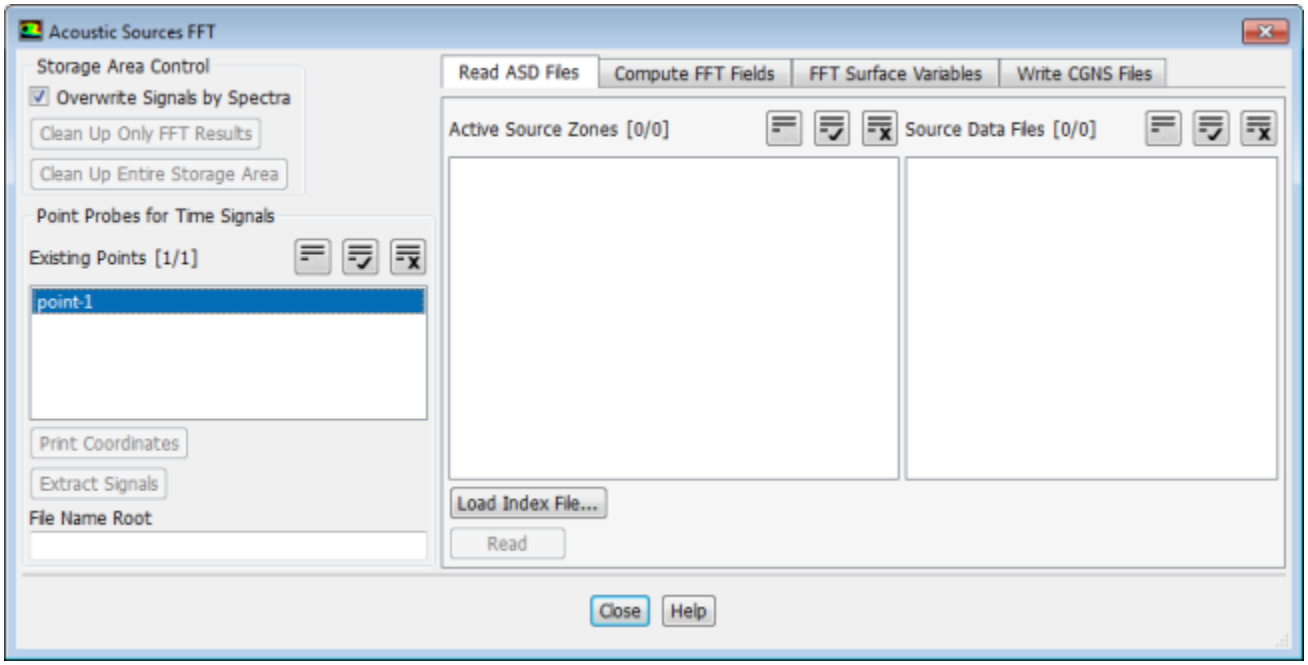
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Compute/Write
computes and saves the sound pressure data. This button will appear only when **Read Unsteady Acoustic Source Data Files** is selected under **Options**.

Receivers...
opens the Acoustic Receivers Dialog Box (p. 2293) in which you can define additional receivers.

37.17.7. Acoustic Sources FFT Dialog Box

The **Acoustic Sources FFT** dialog box allows you to: read pressure signals from the acoustic source data (ASD) files (see Writing Source Data Files (p. 1211)); compute Fourier spectra of these signals at the selected point probes and over the entire source zones; create surface variables for visualizing the spectral properties of the flow pressure signals; and write CGNS files of the spectrum data. See FFT of Acoustic Sources: Band Analysis and Export of Surface Pressure Spectra (p. 1222) for details about the items below.



Controls

Storage Area Control
allows you to specify how memory is allocated in the "storage area" (that is, the large memory array that stores the fields of the pressure histories and the computed fields of the Fourier spectra) when you compute the FFT fields in the **Compute FFT Fields** tab.

Overwrite Signals by Spectra
specifies that the Fourier spectra displace in memory the original time signals, so that they are not kept after the FFT has been computed.

Clean Up Only FFT Results
de-allocates the storage area—so that you can re-compute the FFT without re-reading the pressure histories—and deletes all variables created through the **FFT Surface Variables** tab. This button is only available when the **Overwrite Signals by Spectra** option is disabled.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clean Up Entire Storage Area
de-allocates the storage area—so that you can re-read ASD files and re-compute the FFT with a clipped time range or different window function—and deletes all variables created through the **FFT Surface Variables** tab.

Point Probes for Time Signals
allows you to get information about existing point probes and create pressure history files.

Existing Points
lists all of the point probes created through the Point Surface Dialog Box (p. 2780) that are available for selection.

Print Coordinates
prints in the console the coordinates of the point probe(s) selected in the **Existing Points** selection list.

Extract Signals
creates ASCII files that contain the pressure histories of the point probe(s) selected in the **Existing Points** selection list.

File Name Root
specifies a prefix for the names of the files created by the **Extract Signals** button.

Read ASD Files
allows you to read pressure histories from the acoustic source data (ASD) files.

Active Source Zones
is a list of the face zones for which you can read acoustic source data.

Source Data Files
is a list of the ASD files you can read.

Load Index File...
updates the **Active Source Zones** and **Source Data Files** lists by loading an ASCII index file.

Read
reads the pressure signals from the selected items in the **Active Source Zones** and **Source Data Files** lists.

Compute FFT Fields
allows you to compute FFT fields from the pressure signals read through the **Read ASD Files** tab.

Sampling Data
allows you to reduce the time range of the signal data used to compute the FFT fields.

Clip Time to Range
specifies that the time range of the data sampled is reduced to be between the **Min** and **Max**.

Min
specifies the minimum value of the time range of the signal data when the **Clip Time to Range** option is enabled.

Max
specifies the maximum value of the time range of the signal data when the **Clip Time to Range** option is enabled.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Time Step
reports the size of the time step at which data was sampled.

Re-Estimate Spectral Resolution
updates the values that are displayed in the **Spectral Resolution** group box and that are used to compute the FFT fields, based on the settings in the **Sampling Data** group box.

Spectral Resolution
displays the number of samples and the expected spectrum properties for the provided signals.

Number of Samples
reports the total number of samples in the specified time range.

Samples Used for FFT
reports the number of samples that will be used for FFT (see Table 31.2: Numbers of Data Points Supported by the Prime-Factor FFT Algorithm (p. 1875)).

Frequency Min & Step
reports the expected minimum frequency in the spectrum, which is also the expected frequency step.

Frequency Max
reports the expected maximum frequency in the spectrum.

Number of Modes
reports the expected number of Fourier modes in the spectrum.

Window Function
specifies the shape of the window function used to enforce the signal's periodicity. See Windowing (p. 1873) for details. The default is the **Hanning** window.

Compute
computes the FFT fields based on the settings in the **Compute FFT Fields** tab.

FFT Surface Variables
allows you to create new Fluent variables for postprocessing using the spectrum fields computed through the **Compute FFT Fields** tab.

Modes/Frequency Bands
specifies what the created variables will characterize.

Octave Bands
specifies that the variables characterize proportional frequency bands corresponding to the standard technical octaves.

1/3 Octave Bands
specifies that the variables characterize proportional frequency bands corresponding to the standard technical thirds.

Constant Width Bands
specifies that the variables characterize user-defined equidistant frequency bands.

Set of Modes
specifies that the variables characterize the individual Fourier modes.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Octave Central Frequencies
is a list of the octave central frequencies for which you can **Create** Fluent variables for postprocessing. This selection list is available when **Octave Bands** is selected from the **Modes/Frequency Bands** drop-down list.

1/3-Octave Central Frequencies
is a list of the 1/3-octave central frequencies for which you can **Create** Fluent variables for postprocessing. This selection list is available when **1/3-Octave Bands** is selected from the **Modes/Frequency Bands** drop-down list.

Frequency Min
specifies the minimum frequency of the set of modes for which you want to **Create** Fluent variables for postprocessing. This field is available when **Set of Modes** or **Constant Width Bands** is selected from the **Modes/Frequency Bands** drop-down list.

Frequency Max
specifies the maximum frequency of the set of modes for which you want to **Create** Fluent variables for postprocessing. This field is available when **Set of Modes** or **Constant Width Bands** is selected from the **Modes/Frequency Bands** drop-down list.

Number of Modes to Skip
allows you to "thin" or "sample" the modes for which you want to **Create** the Fluent variables for postprocessing. This field is available when **Set of Modes** is selected from the **Modes/Frequency Bands** drop-down list.

Band Width
specifies the width of the constant width bands, for which you want to **Create** the Fluent variables for postprocessing. This field is available when **Constant Width Bands** is selected from the **Modes/Frequency Bands** drop-down list.

Spectrum Property
displays the type of variables created according to the selected choice in the **Modes/Frequency Bands** drop-down list.

Surface Pressure Level
is displayed when frequency band is selected from the **Modes/Frequency Bands** drop-down list, in order to indicate that the created variables will be the surface pressure level (SPL) fields, in decibels.

Real and Imaginary Amplitude Parts
is displayed when **Set of Modes** is selected from the **Modes/Frequency Bands** drop-down list, in order to indicate that the created variables will be the real and imaginary parts of the complex Fourier amplitudes, with a pair of variables per specified mode.

Existing Variables
lists the processed variables that have been created by the **Create** button, and allows you to make selections for deletion using the **Remove Selected Variables** button.

Remove Selected Variables
deletes the processed variables selected in the **Existing Variables** list from the "storage area" (as described previously), so that you can analyze more than the maximum of 20 individual modes or constant width bands that are allowed to exist at any one time.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create
creates new Fluent variables for postprocessing (based on your settings in the **FFT Surface Variables** tab), updates the **Existing Variables** selection list, and (when **Set of Modes** or **Constant Width Bands** is selected from the **Modes/Frequency Bands** drop-down list) prints information in the console.

Write CGNS Files
allows you to write CGNS files of the spectrum data.

Processed Source Zones
is a list of the zones you can export.

Frequency Series
allows you to reduce the exported spectrum data.

Reduce Frequency Series
specifies that the exported spectrum data is reduced to be between the **Min** and **Max**.

Min
specifies the minimum of the frequency range to be exported when the **Reduce Frequency Series** option is enabled.

Max
specifies the maximum of the frequency range to be exported when the **Reduce Frequency Series** option is enabled.

Number of Frequencies to Skip
allows you to "thin" or "sample" the frequencies to be exported.

Number of Frequencies per File
specifies the number of frequencies exported in each file. A value should be chosen based on the **Number of Modes** field of the **Compute FFT Fields** tab, in order to avoid creating either extremely large output files or a large number of very small files.

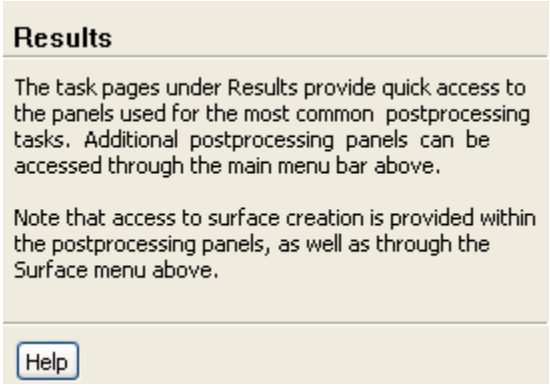
Fourier Transform File Name Root
specifies the root of the names of the output files.

Write
writes CGNS files of the spectrum data.

37.18. Results Task Page

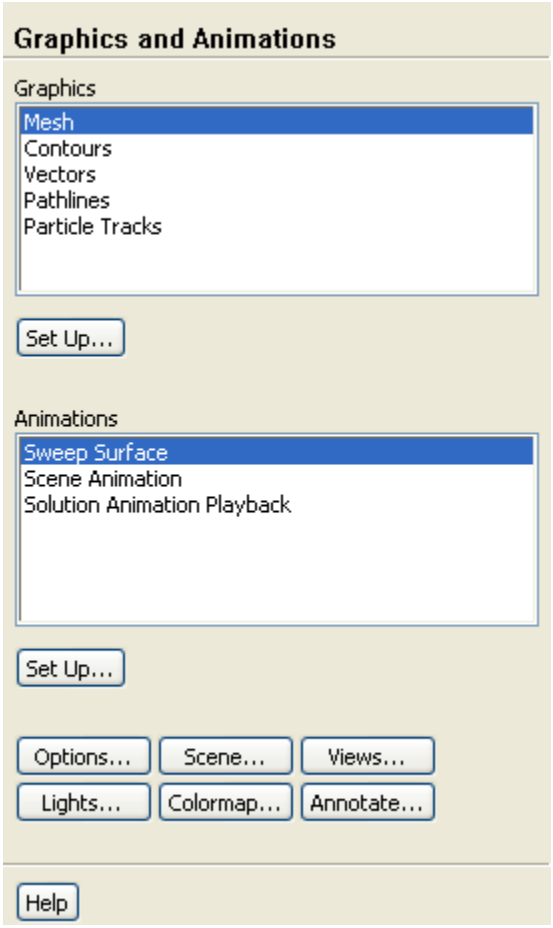
The **Results** task page introduces you to the main tasks involved in setting up and displaying the results of your CFD simulation using ANSYS Fluent.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



37.19. Graphics and Animations Task Page

The **Graphics and Animation** task page allows you to visualize the results of your CFD simulation by allowing you to set up plots of contours, vectors, pathlines, particle tracks, scene descriptions and animations. See Displaying Graphics (p. 1747) for more information.



Controls

Graphics

displays a list of the available graphics objects.

You can double-click an item in the **Graphics** list to open the corresponding dialog box, or you can select the item in the list and click the **Set Up...** button.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mesh
- selecting this item and clicking the **Set Up...** button opens the Mesh Display Dialog Box (p. 2163).

Contours
- selecting this item and clicking the **Set Up...** button opens the Contours Dialog Box (p. 2676).

Vectors
- selecting this item and clicking the **Set Up...** button opens the Vectors Dialog Box (p. 2831).

Pathlines
- selecting this item and clicking the **Set Up...** button opens the Pathlines Dialog Box (p. 2771).

Particle Tracks
- selecting this item and clicking the **Set Up...** button opens the Particle Tracks Dialog Box (p. 2761).

Set Up...
opens the dialog box corresponding to the selected object in the **Graphics** list.

Animations
displays a list of the available animation objects.

You can double-click an item in the **Animations** list to open the corresponding dialog box, or you can select the item in the list and click the **Set Up...** button.

Sweep Surface
- selecting this item and clicking the **Set Up...** button opens the Sweep Surface Dialog Box (p. 2565).

Scene Animation
- selecting this item and clicking the **Set Up...** button opens the Animate Dialog Box (p. 2567).

Solution Animation
- selecting this item and clicking the **Set Up...** button opens the Playback Dialog Box (p. 2571).

Set Up...
opens the dialog box corresponding to the selected object in the **Animations** list.

Options...
opens the Display Options Dialog Box (p. 2573).

Scene...
opens the Scene Description Dialog Box (p. 2577).

Views...
opens the Views Dialog Box (p. 2583).

Lights...
opens the Lights Dialog Box (p. 2588).

Colormap...
opens the Colormap Dialog Box (p. 2589).

Annotate...
opens the Annotate Dialog Box (p. 2592).

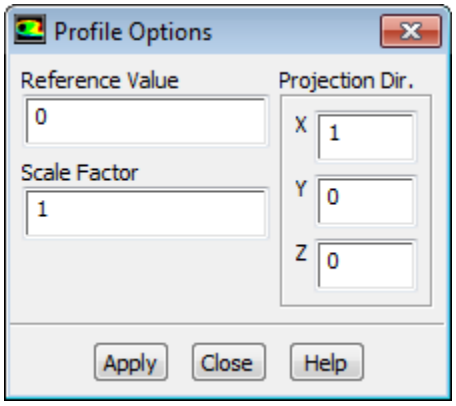
For additional information, see the following sections:

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.19.1. Profile Options Dialog Box
37.19.2. Vector Options Dialog Box
37.19.3. Custom Vectors Dialog Box
37.19.4. Vector Definitions Dialog Box
37.19.5. Path Style Attributes Dialog Box
37.19.6. Ribbon Attributes Dialog Box
37.19.7. Particle Filter Attributes
37.19.8. Reporting Variables Dialog Box
37.19.9. Track Style Attributes Dialog Box
37.19.10. Particle Sphere Style Attributes Dialog Box
37.19.11. Particle Vector Style Attributes Dialog Box
37.19.12. Sweep Surface Dialog Box
37.19.13. Create Surface Dialog Box
37.19.14. Animate Dialog Box
37.19.15. Save Picture Dialog Box
37.19.16. Playback Dialog Box
37.19.17. Display Options Dialog Box
37.19.18. Scene Description Dialog Box
37.19.19. Display Properties Dialog Box
37.19.20. Transformations Dialog Box
37.19.21. Iso-Value Dialog Box
37.19.22. Pathline Attributes Dialog Box
37.19.23. Bounding Frame Dialog Box
37.19.24. Views Dialog Box
37.19.25. Write Views Dialog Box
37.19.26. Mirror Planes Dialog Box
37.19.27. Graphics Periodicity Dialog Box
37.19.28. Camera Parameters Dialog Box
37.19.29. Lights Dialog Box
37.19.30. Colormap Dialog Box
37.19.31. Colormap Editor Dialog Box
37.19.32. Annotate Dialog Box

37.19.1. Profile Options Dialog Box

The **Profile Options** dialog box controls the scaling and projection direction of profiles. It is opened from the Contours Dialog Box (p. 2676), and you will display the profiles using the **Display** button in that dialog box. See Displaying Contours and Profiles (p. 1756) for details about the items below.



Controls

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

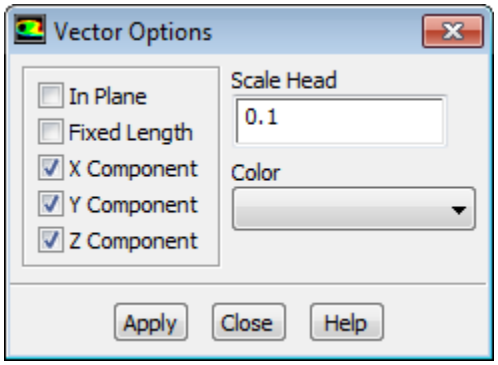
Reference Value
sets the "zero height" reference value for the profile. Any point on the profile with a value equal to the **Reference Value** will be plotted exactly on the defining surface. Values greater than the **Reference Value** will be projected ahead of the surface (in the direction of **Projection Dir.** and scaled by **Scale Factor**), and values less than the **Reference Value** will be projected behind the surface.

Scale Factor
sets the length scale factor for projection. After subtracting off the **Reference Value**, ANSYS Fluent multiplies the resulting solution value by the **Scale Factor** to form a length.

Projection Dir.
sets the direction in which profiles are projected. In 2D, for example, a contour plot of pressure on the entire domain can be projected in the z direction to form a carpet plot, or a contour plot of x velocity on a sequence of x-coordinate slice lines can be projected in the x direction to form a series of velocity profiles.

37.19.2. Vector Options Dialog Box

The **Vector Options** dialog box allows you to set additional parameters for vector displays. It is opened from the Vectors Dialog Box (p. 2831). See Vector Plot Options (p. 1766) for details about the items below.



Controls

In Plane
toggles the display of vector components in the plane of the surface selected for display. This feature is useful for visualizing components that are normal to the flow. See Drawing Vectors in the Plane of the Surface (p. 1767) for details.

Fixed Length
enables the display of vectors that are all the same length. See Displaying Fixed-Length Vectors (p. 1768) for details.

X, Y, Z Component
toggle the display of the Cartesian components of the vectors. See Displaying Vector Components (p. 1768) for details.

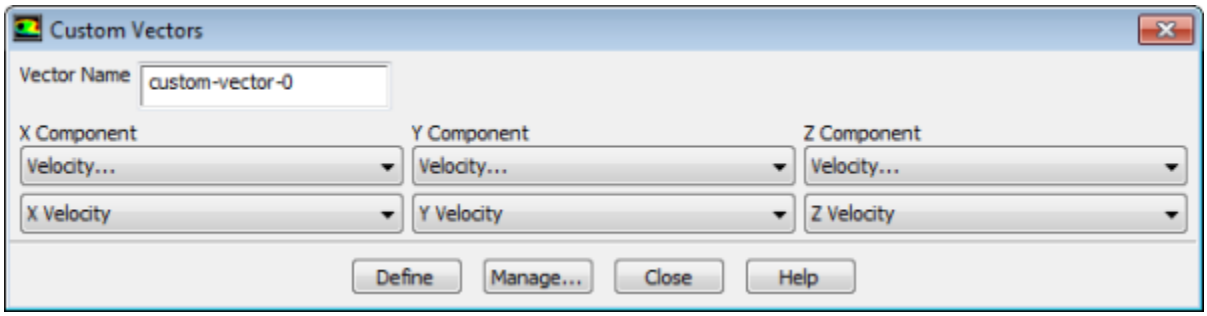
Scale Head
controls the size of the arrowhead on vector styles that include heads.

Color
specifies a single color for the display of all vectors. See Displaying Vectors Using a Single Color (p. 1769) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.19.3. Custom Vectors Dialog Box

The **Custom Vectors** dialog box allows you to define custom vectors based on existing quantities. Any vectors that you define will be added to the **Vectors of** list in the Vectors Dialog Box (p. 2831). To open the **Custom Vectors** dialog box, click **Custom Vectors...** in the **Vectors** dialog box. See Creating and Managing Custom Vectors (p. 1769) for details about custom vectors.



Controls

Vector Name
specifies the name of the vector you are defining. Should you decide to change the name after you have defined the vector, you can do so in the Vector Definitions Dialog Box (p. 2558), which you can open by clicking on the **Manage...** button.

X, Y, Z Component
specify the x, y, and z components of the vector. Each drop-down list contains the available field functions.

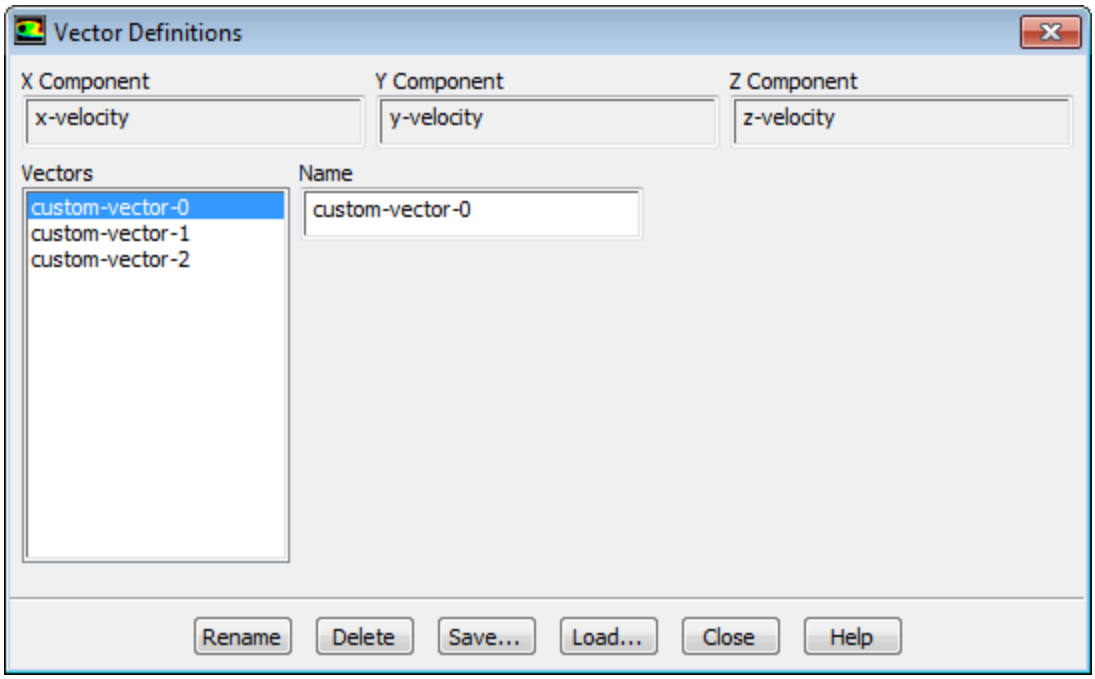
Define
creates the vector and adds it to the **Vectors of** list in the Vectors Dialog Box (p. 2831).

Manage...
opens the Vector Definitions Dialog Box (p. 2558), which enables you to check, rename, save, load, and delete custom vectors.

37.19.4. Vector Definitions Dialog Box

The **Vector Definitions** dialog box allows you to check, rename, save, load, and delete custom vectors that you defined in the Custom Vectors Dialog Box (p. 2558). See Creating and Managing Custom Vectors (p. 1769) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

X, Y, Z Component
display the x, y, and z components of the vector.

Vectors
contains a selectable list of custom vectors. When you select a vector, its components will appear in the **X**, **Y**, and **Z Component** fields, and its name will appear in the **Name** field.

Name
displays the name of the currently selected vector. You can enter a new name in this box if you want to rename the vector.

Rename
changes the name of the selected function to the name specified in the **Name** field.

Delete
deletes the selected vector.

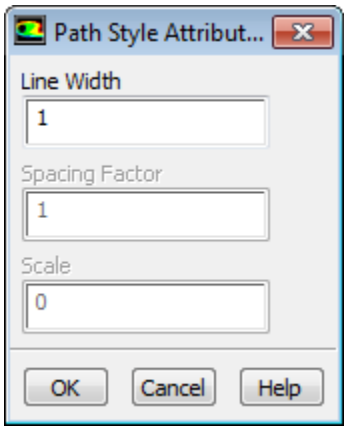
Save...
opens The Select File Dialog Box (p. 19), in which you can specify a file in which to save all of the custom vectors in the **Vectors** list.

Load...
opens the **Select File** dialog box, in which you can specify a file from which to read custom vectors (a file that you saved using the **Save...** button above).

37.19.5. Path Style Attributes Dialog Box

To modify the line width, cylinder radius or marker size, use the **Path Style Attributes** dialog box. You can open this dialog box by clicking the **Attributes...** button in the Pathlines Dialog Box (p. 2771). See Controlling the Pathline Style (p. 1774) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Line Width/Marker Size/Width/
determines the thickness of the pathlines.

Diameter
specifies the diameter of the sphere. This parameter appears only when **sphere** is selected under **Style** in the **Pathlines** dialog box.

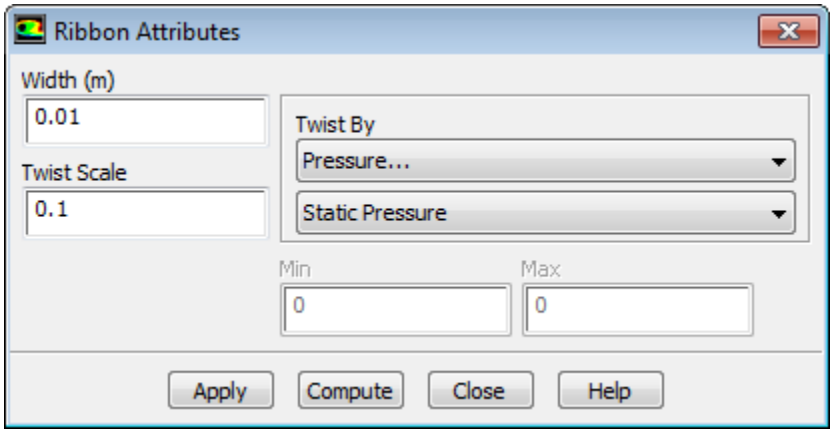
Spacing Factor
controls the spacing of arrows when you use the **line-arrows** style.

Scale
controls the size of the arrow heads when you use the **line-arrows** style.

Detail
specifies the detail applied to the graphical rendering of the spheres. This parameter appears only when **sphere** is selected under **Style** in the **Pathlines** dialog box.

37.19.6. Ribbon Attributes Dialog Box

To modify the ribbon width and set the scalar field by which to twist the ribbon, use the **Ribbon Attributes** dialog box. You can open this dialog box by clicking the **Attributes...** button in the Pathlines Dialog Box (p. 2771) when the selected **Style** is ribbon. See Controlling the Pathline Style (p. 1774) for details about the items below.



Controls

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Width
determines the thickness of the ribbon.

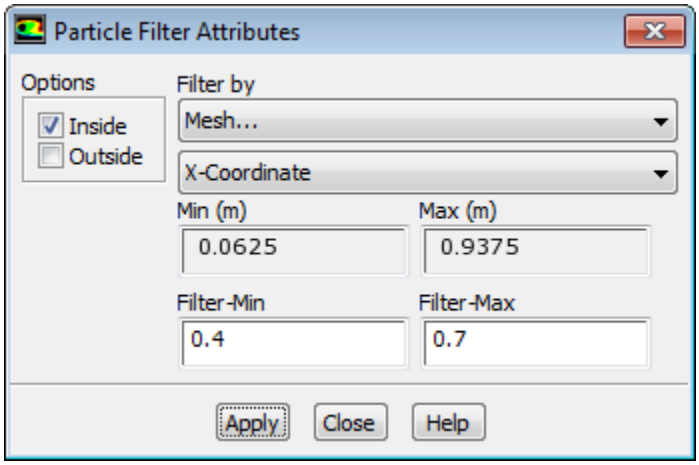
Twist Scale
sets the amount of twist for a given field. To magnify the twist for a field with very little change, increase this factor; to display less twist for a field with dramatic changes, decrease this factor.

Twist By
contains a drop-down list from which you can select a scalar field on which pathline twisting is based.

Min/Max
displays the minimum/maximum value of the scalar field selected in **Twist By**.

37.19.7. Particle Filter Attributes

The **Particle Filter Attributes** dialog box allows you to specify how you would like to filter the particles being displayed. See Particle Filtering (p. 1340) for details about the items below.



Controls

Options
contains the filtering options.

Inside
enables the filtering of particles with values between **Filter-Min** and **Filter-Max**.

Outside
enables the filtering of particles with values less than **Filter-Min** or greater than **Filter-Max**.

Filter by
contains a list from which you can select any field variable, except for **Custom Field Functions...**, to be used as a filter variable.

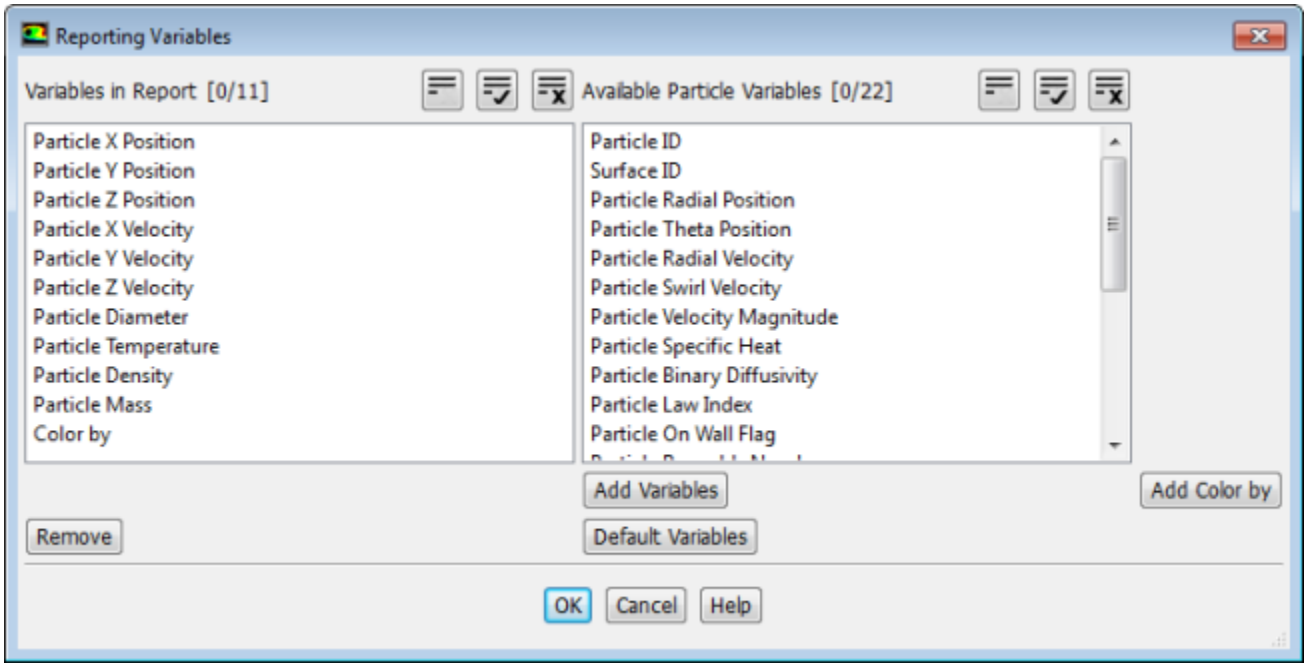
Min/Max
displays the minimum and maximum values of the selected field variable. The real number field values are not editable; they are purely informational.

Filter-Min/Filter-Max
defines the minimum/maximum filter threshold.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.19.8. Reporting Variables Dialog Box

The **Reporting Variables** dialog box allows you to control the particle variables that you include in your reporting. See Step-by-Step Reporting of Trajectories (p. 1347) for details about the items below.



Controls

Variables in Report
contains all variables currently in the report.

Remove
removes the selected variable from the report.

Default Variables
restores the default list.

Available Particle Variables
contains the particle variables that are available for you to select.

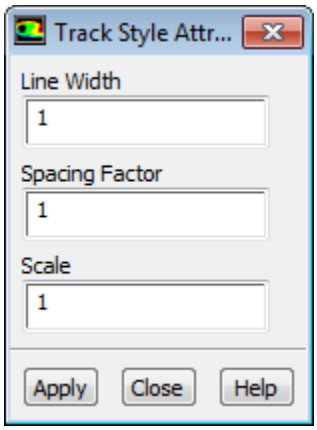
Add Variables
takes the selected variable from the **Available Particle Variables** list and adds it to the **Variables in Report** list.

Add Color By
adds the **Color by** variable to the **Variables in Report** list.

37.19.9. Track Style Attributes Dialog Box

To modify the line width, cylinder radius or marker size, use the **Track Style Attributes** dialog box. You can open this dialog box by clicking the **Attributes...** button in the Particle Tracks Dialog Box (p. 2761). See Controlling the Particle Tracking Style (p. 1332) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

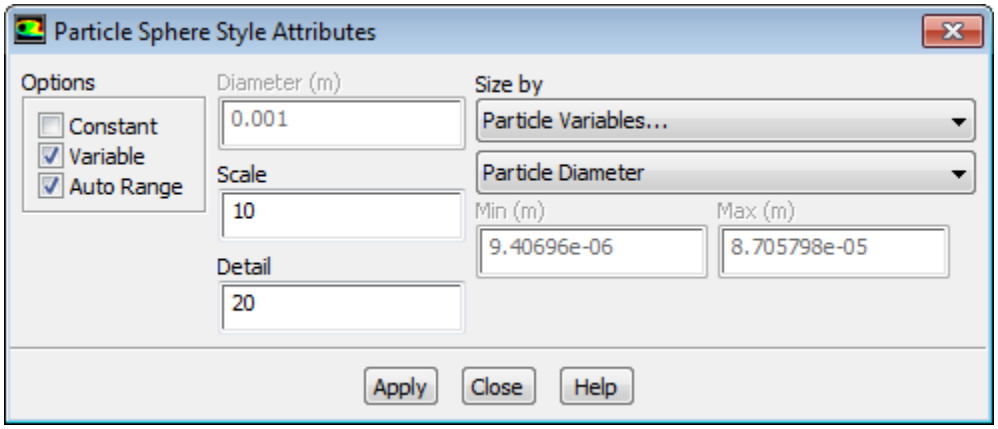
Line Width/Marker Size/Width
determines the thickness of the particle tracks.

Spacing Factor
controls the spacing of arrows when you use the **line-arrows** style.

Scale
controls the size of the arrow heads when you use the **line-arrows** style.

37.19.10. Particle Sphere Style Attributes Dialog Box

To modify the attributes of the particle sphere, use the **Particle Sphere Style Attributes** dialog box. Select **sphere** from the **Track Style** drop-down list and click the **Attributes...** button in the Particle Tracks Dialog Box (p. 2761) to open the **Particle Sphere Style Attributes** dialog box. See Controlling the Particle Tracking Style (p. 1332) for details about the items below.



Controls

Options
allows you to choose how you would like to specify the particle diameter.

Constant
allows you to specify the diameter as a constant value.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Variable
allows you to select a particle variable to estimate the size of the spheres.

Auto Range
when disabled clips the displayed particles to the values given in **Min** and **Max**.

Diameter
specifies the diameter of the sphere.

Scale
allows you to scale the spheres by the factor entered in this field.

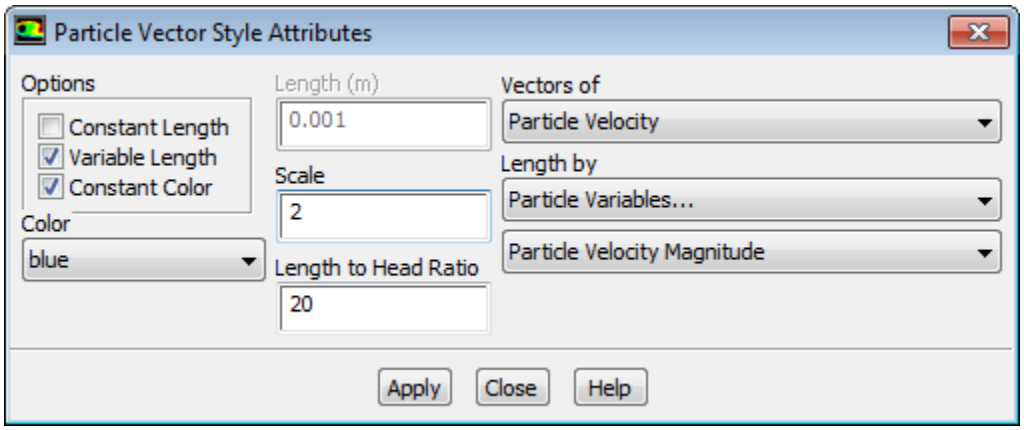
Detail
specifies the detail applied to the graphical rendering of the spheres.

Size by
contains a list of variables by which you can estimate the size of your particle. This list is selectable only when you are using the **Variable** option.

Min/Max
defines the minimum and maximum values of the selected field variable to display.

37.19.11. Particle Vector Style Attributes Dialog Box

To modify the attributes of the vector styles, use the **Particle Vector Style Attributes** dialog box. See Controlling the Vector Style of Particle Tracks (p. 1334) for details about the items below.



Controls

Options
allows you to choose how you would like to specify the particle diameter.

Constant Length
allows you to specify the vector length as a constant value.

Variable Length
results in a vector length that is based on the variable selected under **Length by**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Constant Color
when enabled allows you to select a vector color from the **Color** drop-down list. Otherwise, the vector is colored based on the variable selected in the **Particle Tracks** dialog box (seen in the **Mesh Colors** dialog box when **Draw Mesh** is enabled).

Length
specifies the length of the vector.

Scale
allows you to scale the vectors by the factor entered in this field.

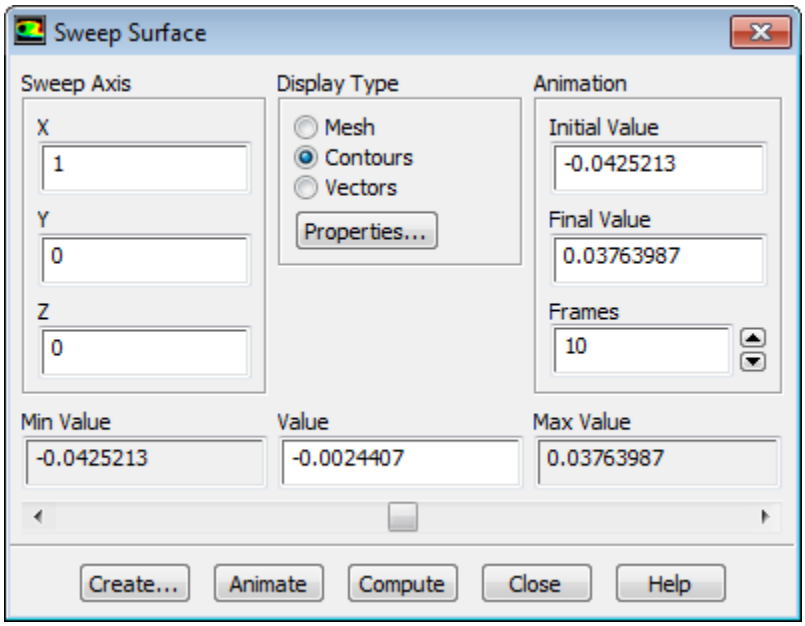
Length to Head Ratio
is the ratio of vector length to vector head size.

Vectors of
contains the particle vector variable to display.

Length by
is used to estimate the length of the vector when the **Variable Length** option is enabled.

37.19.12. Sweep Surface Dialog Box

The **Sweep Surface** dialog box controls the display and animation of mesh, contour, and vector plots generated on a sweep surface. See Displaying Results on a Sweep Surface (p. 1782) for details about the items below.



Controls

Sweep Axis
specifies the (**X**, **Y**, **Z**) vector representing the axis along which the surface should be swept.

Display Type
specifies the type of display to be swept through the domain (**Mesh**, **Contours**, or **Vectors**).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Properties...
opens the Contours Dialog Box (p. 2676) if **Contours** is the selected **Display Type**, or the Vectors Dialog Box (p. 2831) if **Vectors** is the selected **Display Type**. (This button is not available if **Mesh** is the selected **Display Type**.)

Animation
contains controls for animating the sweep-surface display.

Initial Value, Final Value
specify the initial and final positions for the animation.

Frames
specifies the number of frames in the animation.

Min Value, Max Value
show the minimum and maximum extents of the domain along the specified **Sweep Axis**. These values are updated when you click **Compute**.

Value
shows the current position at which the requested display is plotted. You can change the value by moving the slide bar below it, or by entering a new value and pressing the <RETURN> key.

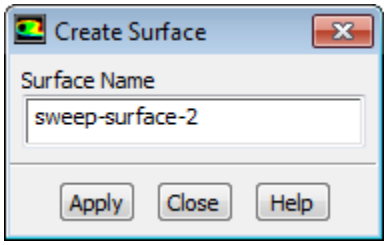
Create...
opens the Create Surface Dialog Box (p. 2566), where you can create a surface from the currently-displayed sweep surface.

Animate
animates the display, sweeping the requested display through the domain along the specified axis.

Compute
updates the **Min Value** and **Max Value** to reflect the minimum and maximum extents of the domain along the specified **Sweep Axis**.

37.19.13. Create Surface Dialog Box

The **Create Surface** dialog box allows you to save a sweep surface for later use. You can open it by clicking **Create...** in the Sweep Surface Dialog Box (p. 2565). See Displaying Results on a Sweep Surface (p. 1782) for details about the items below.



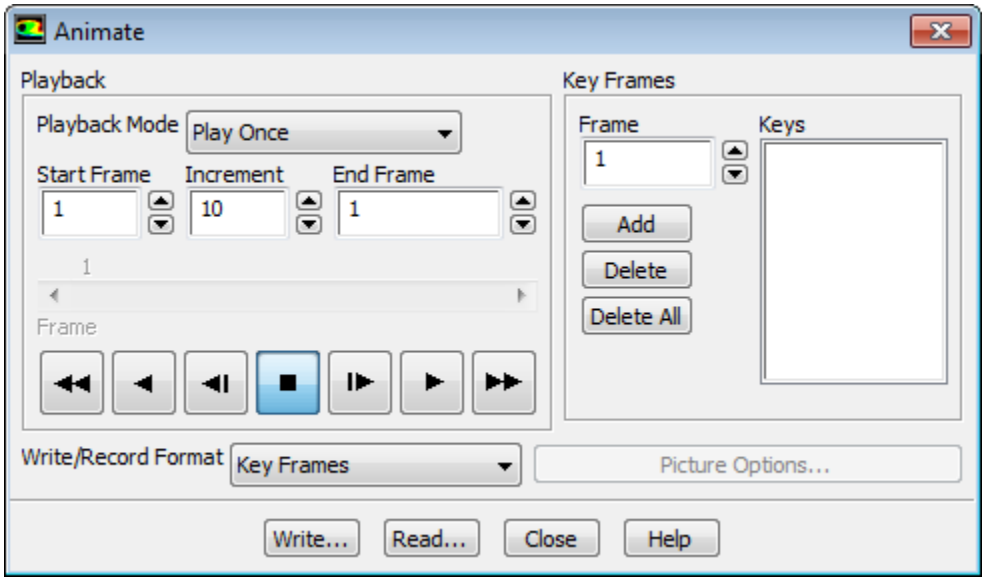
Controls

Surface Name
specifies a name for the surface to be created.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.19.14. Animate Dialog Box

The **Animate** dialog box allows you to specify key frames that define the basic movement of an animated sequence, and then play back the animation. ANSYS Fluent interpolates between your specified key frames. See Animating Graphics (p. 1825) for details about the items below.



Controls

Playback
contains the controls that you use to play back the animation. See Playing an Animation (p. 1826) for details.

Playback Mode
contains a drop-down list of playback options.

Play Once
sets the option to play back frames from **Start Frame** to **End Frame** once.

Auto Repeat
sets the option to continually play back frames from **Start Frame** to **End Frame**.

Auto Reverse
sets the option to continually play back the images while reversing playback direction after each set.

Start Frame, End Frame
set the frames at which the animation should begin and end. By changing these numbers you can view a subset of the frames.

Increment
sets the number of frames to increment the frame-counter by when you use the fast-forward or fast-reverse buttons.

Frame
shows the number of the frame that is currently displayed, as well as its relative position in the entire animation. If you slide the bar to a different location, the frame corresponding to the new frame number will be displayed in the graphics window.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(Tape Player Buttons)
allow you to play the animation forward and backward, fast-reverse and fast-forward the animation, and stop it. The buttons function in a way similar to those on a standard video cassette player.

Key Frames
contains the controls that you use to define the key frames for the animation. See Creating an Animation (p. 1826) for details.

Frame
sets the number to be assigned to the next key frame added to the list of **Keys**.

Keys
contains a list of the key frames that have been defined. If you select a key frame in this list, the associated scene will be displayed in the graphics window.

Add
creates a key frame with the number shown in **Frame** for the scene currently displayed in the graphics window.

Important

Be sure to change the frame number before you add the new key frame so that you will not overwrite the last key frame that you created.

Delete
deletes the key frame that is selected in the **Keys** list.

Delete All
deletes all key frames in the **Keys** list.

Write/Record Format
specifies **Key Frames**, **Picture Files**, **MPEG**, or **Video** (not available on Windows) as the format in which to save the animation. See Saving an Animation (p. 1828) for details about these options.

Picture Options...
opens the Save Picture Dialog Box (p. 2568), in which you can specify parameters for saving the animation to picture files. This button is available only when **Picture Files** is selected as the **Write/Record Format**.

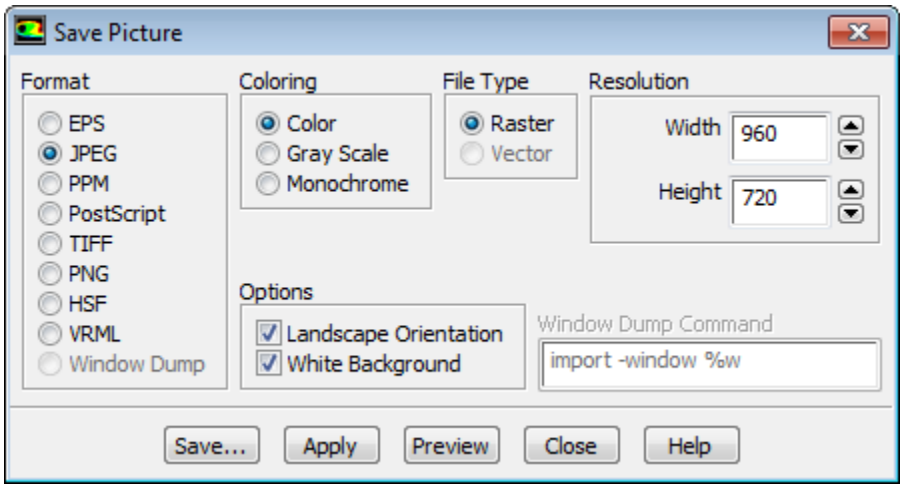
Write...
opens The Select File Dialog Box (p. 19), in which you can specify a name for the animation file and save it.

Read...
opens the **Select File** dialog box, in which you can specify the name of the animation file to be read. Note that the current case and data should contain the surfaces and any other information that the key frame description refers to. See Reading an Animation File (p. 1829).

37.19.15. Save Picture Dialog Box

The **Save Picture** dialog box allows you to set save picture parameters and save picture files of graphics windows. See Saving Picture Files (p. 102) for details on the use of this dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Format

allows you to select the format of picture files.

EPS

(Encapsulated PostScript) output is the same as PostScript output, with the addition of Adobe Document Structuring Conventions (v2) statements. Currently, no preview bitmap is included in EPS output. Often, programs that import EPS files use the preview bitmap to display on-screen, although the actual vector PostScript information is used for printing (on a PostScript device). You can save EPS files in raster or vector format.

JPEG

is a common raster file format.

PPM

output is a common raster file format.

PostScript

is a common vector file format. You can also choose to save a PostScript file in raster format.

TIFF

is a common raster file format.

PNG

is a common raster file format.

HSF

is HOOPS Visualize Stream Format, a highly-compressible and streamable 2D/3D file format.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VRML
is a graphics interchange format that allows export of 3D geometrical entities that you can display in the ANSYS Fluent graphics window. This format can commonly be used by VR systems and in particular the 3D geometry can be viewed and manipulated in a web-browser graphics window.

Important

Non-geometric entities such as text, titles, color bars, and orientation axis are not exported. In addition, most display or visibility characteristics set in ANSYS Fluent, such as lighting, shading method, transparency, face and edge visibility, outer face culling, and hidden line removal, are not explicitly exported but are controlled by the software used to view the VRML file.

Window Dump
(Linux systems only) selects a window dump operation for generating the picture. With this format, you will need to specify the appropriate **Window Dump Command**.

Coloring
(all formats except **Window Dump**) specifies the color mode for the picture file.

Color
specifies a color-scale copy.

Gray Scale
specifies a gray-scale copy.

Monochrome
specifies a black-and-white copy.

Important

Most monochrome PostScript devices will render **Color** images in shades of gray, but to ensure that the color ramp is rendered as a linearly-increasing gray ramp, you should select **Gray Scale**.

File Type
specifies the type of picture file to be saved. See Choosing the File Type (p. 105) for details.

Raster
specifies a raster type picture. The supported raster formats are **EPS**, **JPEG**, **PPM**, **PostScript**, **TIFF**, and **PNG**.

Vector
specifies a vector type picture. The supported vector formats are **EPS**, **PostScript**, and **VRML**.

Resolution
specifies the resolution or the size (in pixels) of the picture.

DPI
specifies the resolution of **EPS** and **PostScript** files in dots per inch (DPI). The default value for **DPI** is set to 75.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Width
specifies the width of the raster picture image.

Height
specifies the height of the raster picture image.

*The default value for **Width** and **Height** is set to zero, so that the default picture is generated at the same resolution as the active graphics window.*

Options
contains additional options for all picture formats except **Window Dump**.

Landscape Orientation
specifies the orientation of the picture. If this option is enabled, the picture is made in landscape mode; otherwise, it is made in portrait mode.

White Background
controls the foreground/background color. If this option is enabled, the foreground and background colors of graphics windows being saved as pictures will be swapped. Hence, it allows you to save pictures with a white background and a black foreground, while the graphics windows are displayed with a black background and white foreground.

Window Dump Command
(Linux systems only) specifies the command to be used to save the picture file, when you select the **Window Dump** format. See Window Dumps (Linux Systems Only) (p. 106) for details.

Save...
opens The Select File Dialog Box (p. 19), in which you can specify a name for the picture file to be saved and then save the file. The resulting file will contain a picture of the active graphics window.

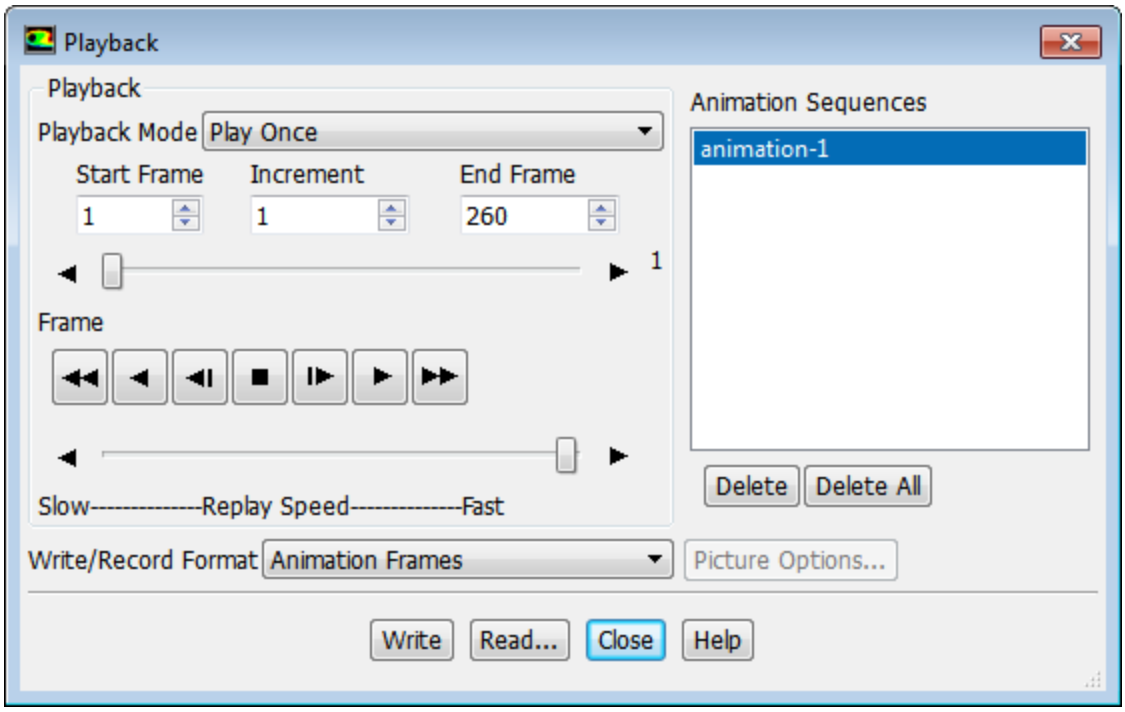
Apply
saves the current settings. ANSYS Fluent will use these settings when making subsequent pictures.

Preview
applies the current settings to the active graphics window so that you can investigate the effects of different options interactively before saving the final picture.

37.19.16. Playback Dialog Box

The **Playback** dialog box allows you to play back an animation sequence. See Playing an Animation Sequence (p. 1650) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Playback
contains the controls that you use to play back the selected animation sequence.

Playback Mode
contains a drop-down list of playback options.

Play Once
sets the option to play back frames from **Start Frame** to **End Frame** once.

Auto Repeat
sets the option to continually play back frames from **Start Frame** to **End Frame**.

Auto Reverse
sets the option to continually play back the images while reversing playback direction after each set.

Start Frame, End Frame
set the frames at which the animation should begin and end. By changing these numbers you can view a subset of the frames.

Increment
sets the number of frames to increment the frame-counter by when you use the fast-forward or fast-reverse buttons.

Frame
shows the number of the frame that is currently displayed, as well as its relative position in the entire animation. If you slide the bar to a different location, the frame corresponding to the new frame number will be displayed in the graphics window.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(Tape Player Buttons)
allow you to play the animation forward and backward, fast-reverse and fast-forward the animation, and stop it. The buttons function in a way similar to those on a standard video cassette player.

Replay Speed
controls the playback speed for the animation. Move the **Replay Speed** slider bar to the left to reduce the playback speed (and to the right to increase it).

Animation Sequences
contains the controls that you use to define the sequence to be played back.

Sequences
contains a list of the animation sequences that have been defined.

Delete
deletes the animation sequence that is selected in the **Sequences** list.

Delete All
deletes all animation sequences in the **Sequences** list.

Write/Record Format
specifies **Animation Frames**, **Picture Files**, or **MPEG** as the format in which to save the animation. See Saving an Animation Sequence (p. 1652) for details about these options.

Picture Options...
opens the Save Picture Dialog Box (p. 2568), in which you can specify parameters for saving the animation to picture files. This button is available only when **Picture Files** is selected as the **Write/Record Format**.

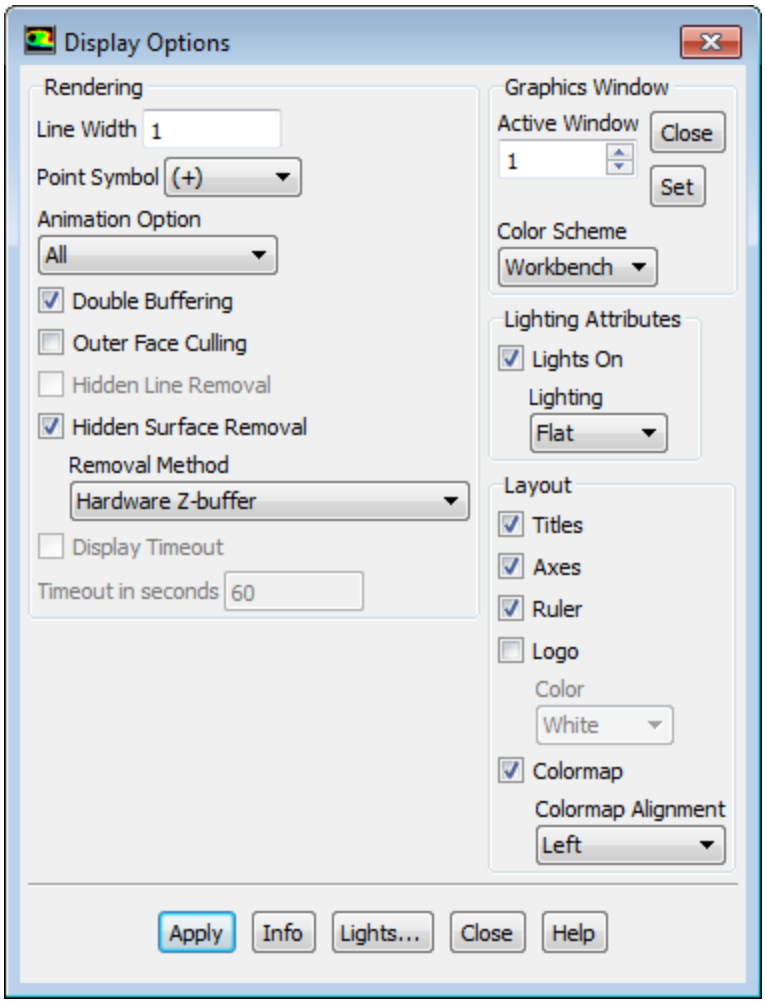
Write
saves the specified file(s) in the current working directory.

Read...
opens The Select File Dialog Box (p. 19), in which you can specify the name of the solution animation file to be read. See Reading an Animation Sequence (p. 1654) for details.

37.19.17. Display Options Dialog Box

The **Display Options** dialog box provides an interactive mechanism for setting attributes or options that control how and where a scene is rendered.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Rendering

allows you to modify characteristics of the display that are related to the way in which scenes are rendered. See Modifying the Rendering Options (p. 1798) for details about these items.

Line Width

controls the thickness of lines. The default is 1.

Point Symbol

sets the symbol used for nodes and data points.

Animation Option

contains a drop-down list of animation options: **All** and **Wireframe**. **Wireframe** uses a wireframe representation of all geometry during mouse manipulation This option is turned on by default. You should turn it off only if your computer has a graphics accelerator; otherwise the mouse manipulation may be very slow.

Double Buffering

turns double buffering on or off, if it is supported by the driver. Double buffering dramatically reduces screen flicker during graphics updates. Note that if your display hardware does not support double buffering and you turn this option on, double buffering will be done in software. Software double buffering uses extra memory.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Outer Face Culling
allows you to turn off the display of outer faces in wall zones. **Outer Face Culling** is useful for displaying both sides of a slit wall. By default, when you display a slit wall, one side will "bleed" through to the other. When you turn on the **Outer Face Culling** option, the display of a slit wall will show each side distinctly as you rotate the display. This option can also be useful for displaying two-sided walls (walls with fluid or solid cells on both sides).

Hidden Line Removal
turns hidden line removal on or off. If you do not use hidden line removal, ANSYS Fluent will not try to determine which lines in the display are behind others; it will display all of them, and a cluttered display will result. You should turn this option off if you are working with a 2D problem or with geometries that do not overlap. Note that this option is not available when using the **Workbench Color Scheme**.

Hidden Surface Removal
turns hidden surface removal on or off. If you do not use hidden surface removal, ANSYS Fluent will not try to determine which surfaces in the display are behind others; it will display all of them, and a cluttered display will result. You should turn this option off if you are working with a 2D problem or with geometries that do not overlap.

Removal Method
chooses the method to be used for hidden surface removal. These options vary in speed and quality, depending on the device you are using. The choices are listed below.

Hardware Z-buffer
is the fastest method if your hardware supports it. The accuracy and speed of this method is hardware dependent.

Painters
will show fewer edge-aliasing effects than hardware-z-buffer. This method is often used instead of software-z-buffer when memory is limited.

Software Z-buffer
is the fastest of the accurate software methods available (especially for complex scenes), but it is memory intensive.

Z-sort only
is a fast software method, but it is not as accurate as software-z-buffer.

Timeout in seconds
specifies the value for the timeout.

Graphics Window
allows you to open and close graphics windows. See Opening Multiple Graphics Windows (p. 1786) for details.

Active Window
indicates the graphics window to be opened, closed, or set active. (A graphics window's ID is displayed in its title.)

Open/Close
opens or closes the window with the ID shown in the **Active Window** box. If the indicated window is open, the **Close** button will appear, and if the indicated window is not open, the **Open** button will appear.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Set
sets the window with the ID shown in the **Active Window** box to be the active graphics window. See Setting the Active Window (p. 1787) for details.

Color Scheme
contains a drop-down list of available graphics window color schemes to choose from. Choices are **Workbench** (blue background) and **Classic** (black background).

Lighting Attributes
controls lighting attributes for all lights in the active graphics window. See Adding Lights (p. 1796) for details.

Lights On
turns *all* lights in the active graphics window on or off.

Lighting
specifies the method to be used in lighting interpolation: **Flat**, **Gouraud**, or **Phong**. (**Flat** is the most basic method: there is no interpolation within the individual polygonal facets. **Gouraud** and **Phong** have smoother gradations of color because they interpolate on each facet.)

Layout
controls the display of captions, axes and the colormap in the graphics display window. See Changing the Legend Display (p. 1788) for details.

Titles
enables/disables the display of all captions in the graphics display window.

Axes
enables/disables the display of the axis triad.

Ruler
enables/disables the display of the ruler.

Logo
allows you to hide or display the ANSYS logo.

Color
contains a drop-down list of colors that can be used for the logo. The choices are **White** (default) or **Black**.

Colormap
enables/disables the display of the color scale.

Colormap Alignment
allows you to adjust the alignment of colormap in the graphics display. Select the side (**Top**, **Bottom**, **Left**, and **Right**) from the drop-down list, where you want to align the colormap.

Apply
applies the specified attributes and re-renders the scene in the active graphics window with the new attributes. To see the effect of the new attributes on other graphics windows, you must redisplay them.

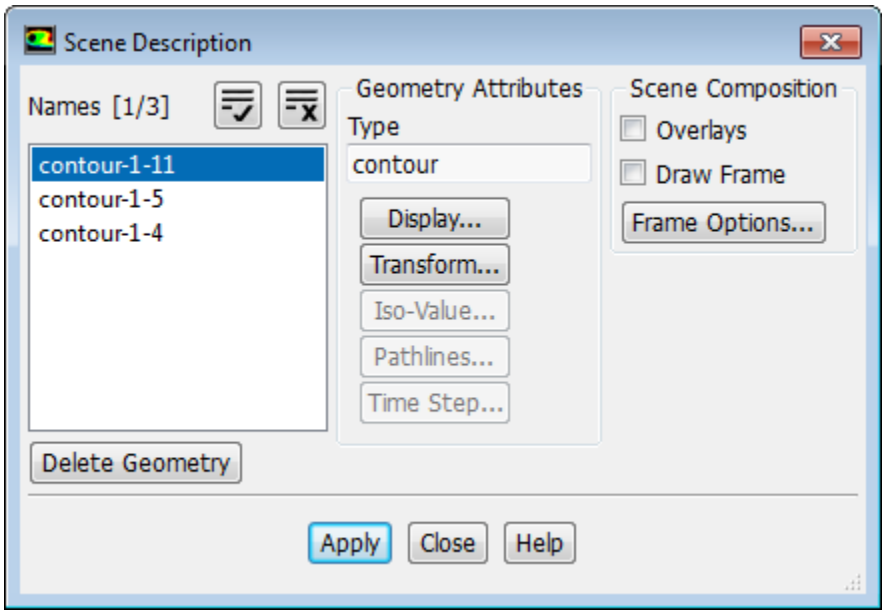
Info
prints out information about your graphics driver in the console.

Lights...
opens the Lights Dialog Box (p. 2588), which allows you to create, delete, and modify directional light sources.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.19.18. Scene Description Dialog Box

The **Scene Description** dialog box allows you to turn overlays on and off, select geometric objects in the display for modification or deletion, and open dialog boxes that control various characteristics of the selected object(s). (Note that you cannot use the **Scene Description** dialog box to control XY plot and histogram displays.) See Advanced Scene Composition (p. 1815) for details about the items below.



Controls

Names

contains a list of the geometric objects that currently exist in the scene (including those that are presently invisible). You can specify the object or objects to be manipulated by selecting names in this list. If you select more than one object at a time, any operation (transformation, color specification, and so on) will apply to all the selected objects. You can also select objects by clicking on them in the graphics display using the mouse probe button, which is, by default, the right mouse button. (See Controlling the Mouse Button Functions (p. 1800) for information about mouse button functions.) To deselect a selected object, simply click its name in the **Names** list. See Selecting the Object(s) to be Manipulated (p. 1817) for details.

Geometry Attributes

contains information about the type of the selected geometric object and push buttons that open dialog boxes for modifying the object.

Type

reports the type of the selected object. Possible types include `mesh`, `surface`, `contour`, `vector`, and `Group`. This information is especially helpful when you need to distinguish two or more objects with the same name. When more than one object is selected, the type displayed is `Group`.

Display...

opens the Display Properties Dialog Box (p. 2578), which allows you to change the color, visibility, and other properties for the selected object.

Transform...

opens the Transformations Dialog Box (p. 2580), which allows you to translate, rotate, and scale the selected object.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Iso-Value...
opens the Iso-Value Dialog Box (p. 2581), which allows you to change the isovalue of an isosurface. This push button is available only if the geometric object selected in the **Names** list is an isosurface or an object on an isosurface (contour on an isosurface, for example); otherwise it is grayed out.

Pathlines...
opens the Pathline Attributes Dialog Box (p. 2582), which allows you to set the maximum number of steps for the selected pathlines. This is most useful in animating the path of a set of particles.

Scene Composition
contains controls for enabling overlays and bounding frames.

Overlays
activates the superimposition of a new geometry onto a currently displayed geometry. See Advanced Graphics Overlays (p. 1785) for details.

Draw Frame
activates the display of a bounding frame in the graphics display. See Adding a Bounding Frame (p. 1823) for details.

Frame Options...
opens the Bounding Frame Dialog Box (p. 2582), in which you can define properties of the bounding frame. See Adding a Bounding Frame (p. 1823) for details.

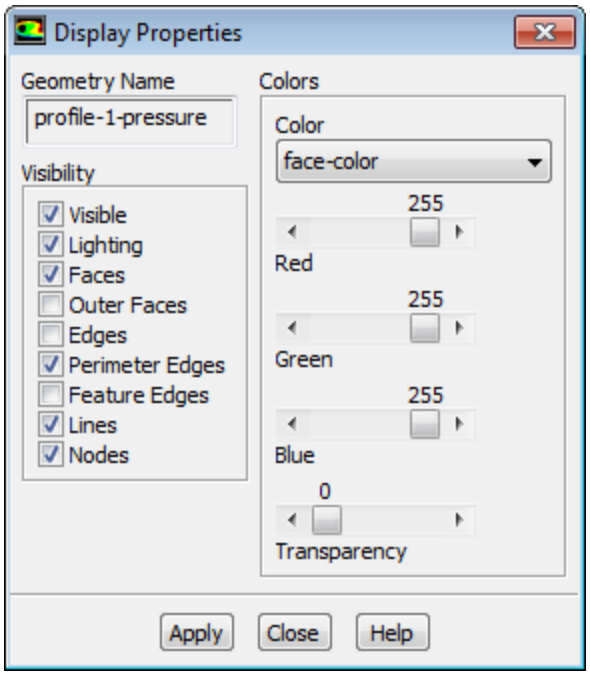
Delete Geometry
deletes the geometric object that is currently selected in the **Names** list. The ability to delete individual objects is especially useful if you have overlays on and you generate an unwanted object (for example, if you generate contours of the wrong variable). You can simply delete the unwanted object and continue your scene composition, instead of starting over from the beginning.

Apply
saves the status of **Overlays** and **Draw Frame**. When you turn **Overlays** or **Draw Frame** on or off, you must click the **Apply** button to see the effect of the change on subsequent display operations.

37.19.19. Display Properties Dialog Box

To modify the color, visibility, and other display properties for individual geometric objects in the graphics display, use the **Display Properties** dialog box. You can open this dialog box by clicking the **Display...** push button in the Scene Description Dialog Box (p. 2577). See Changing an Object's Display Properties (p. 1817) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Geometry Name
displays the name of the object you selected for modification in the Scene Description Dialog Box (p. 2577).

Visibility
contains check buttons that control options related to the visibility of the selected object. See Controlling Visibility (p. 1818) for details.

Visible
toggles the visibility of the selected object. If it is turned on, the object will be visible in the display, and if it is turned off, the object will be invisible.

Lighting
turns the effect of lighting for the selected object on or off. You can choose to have lighting affect only certain objects instead of all of them. Note that if **Lighting** is turned on for an object such as a contour or vector plot, the colors in the plot will not be exactly the same as those in the colormap at the left of the display.

Faces
toggles the filled display of faces for the selected mesh or surface object. Turning **Faces** on has the same effect as turning on the display of faces in the Mesh Display Dialog Box (p. 2163).

Outer Faces
toggles the display of outer faces.

Edges
toggles the display of interior and exterior edges of the geometric object.

Perimeter Edges
toggles the display of the outline of the geometric object. (This option has no effect on the display of meshes.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Feature Edges
toggles the display of feature lines (if any) of the geometric object.

Lines
toggles the display of the lines (if any) in the geometric object.

Nodes
toggles the display of the nodes (if any) in the geometric object.

Colors
contains controls for setting face, edge, line, and node colors, and transparency for faces. See Controlling Object Color and Transparency (p. 1818) for details.

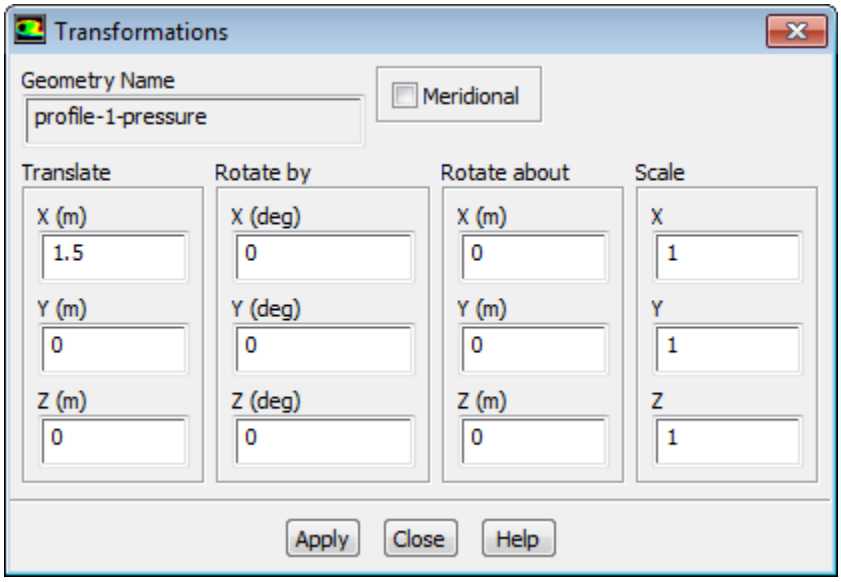
Color
specifies the face, edge, line, or node color for modification. When you turn on this button, the color scales below will show the current color specification, which you can modify by moving the sliders on the color scales.

Red, Green, Blue
are color scales with which you can specify the RGB components of the face, edge or line color.

Transparency
sets the relative transparency of the selected object. An object with a transparency of 0 is opaque, and an object with a transparency of 100 is transparent.

37.19.20. Transformations Dialog Box

You can use the **Transformations** dialog box to translate, rotate, or scale individual objects in the graphics display. To open this dialog box, click the **Transform...** push button in the Scene Description Dialog Box (p. 2577). See Transforming Geometric Objects in a Scene (p. 1819) for details about the items below.



Controls

Geometry Name
displays the name of the object you selected for modification in the Scene Description Dialog Box (p. 2577).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Meridional
(3D only) enables the display of the meridional view. This option is especially useful in turbomachinery applications.

Translate
contains **X**, **Y**, and **Z** real number fields in which you can enter the distance by which to translate the selected object in each direction.

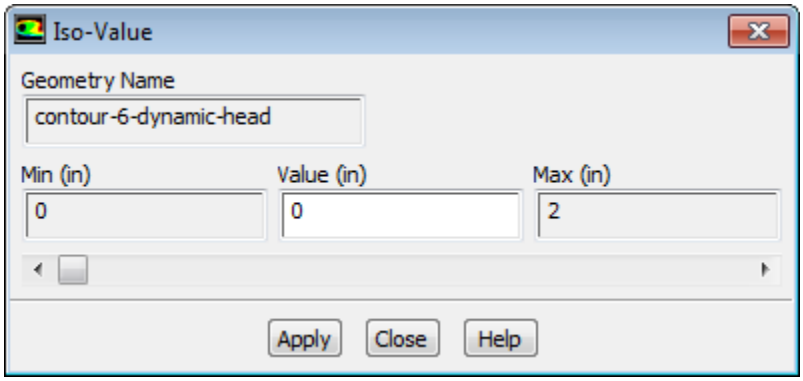
Rotate by
contains **X**, **Y**,and **Z** integer number fields in which you can enter the number of degrees by which to rotate the selected object about each axis.

Rotate about
specifies the point about which to rotate the object.

Scale
contains **X**, **Y**, and **Z** real number fields in which you can enter the amount by which to scale the selected object in each direction. To avoid distortion of the object's shape, be sure to specify the same value for all three entries.

37.19.21. Iso-Value Dialog Box

The **Iso-Value** dialog box allows you to change the isovalue of an isosurface. The isosurface can be selected directly in the **Names** list or indirectly by selecting an object displayed on the isosurface. When you change the isovalue, any contours, vectors, and so on that were displayed on the original isosurface will be displayed on the isosurface with the new isovalue. To open this dialog box, click the **Iso-Value...** push button in the Scene Description Dialog Box (p. 2577). See Modifying Iso-Values (p. 1821) for details about using this dialog box.



Controls

Geometry Name
displays the name of the geometric object (isosurface) you selected for modification in the Scene Description Dialog Box (p. 2577).

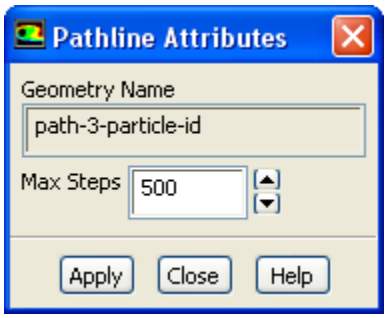
Min, Max
show the minimum and maximum values of the isosurface variable.

Value
sets the new isovalue for isosurfaces. After you change the value and click **Apply**, contours, vectors, or pathlines that were displayed on the original isosurface will be displayed for the new isovalue. You can also use the slide bar to change the **Value**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.19.22. Pathline Attributes Dialog Box

The **Pathline Attributes** dialog box allows you to change the number of steps used in the computation of pathlines. This is most useful in creating animations of pathlines. To open this dialog box, click the **Pathlines...** button in the Scene Description Dialog Box (p. 2577). See Modifying Pathline Attributes (p. 1822) for details about using this dialog box.



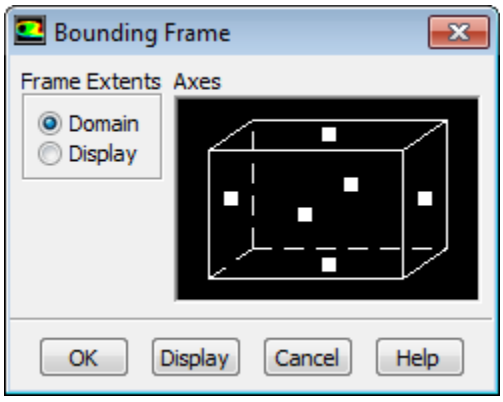
Controls

Geometry Name
displays the name of the geometric object you selected for modification in the Scene Description Dialog Box (p. 2577).

Max Steps
sets the new maximum number of steps for pathline computation. After you change the value and click **Apply**, the selected pathline will be recomputed and redrawn.

37.19.23. Bounding Frame Dialog Box

The **Bounding Frame** dialog box allows you to add a bounding frame with optional measure markings to the display of the domain. See Adding a Bounding Frame (p. 1823) for details about the items in this dialog box.



Controls

Frame Extents
indicates the extents of the bounding frame.

Domain
specifies that the frame should encompass the domain extents.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

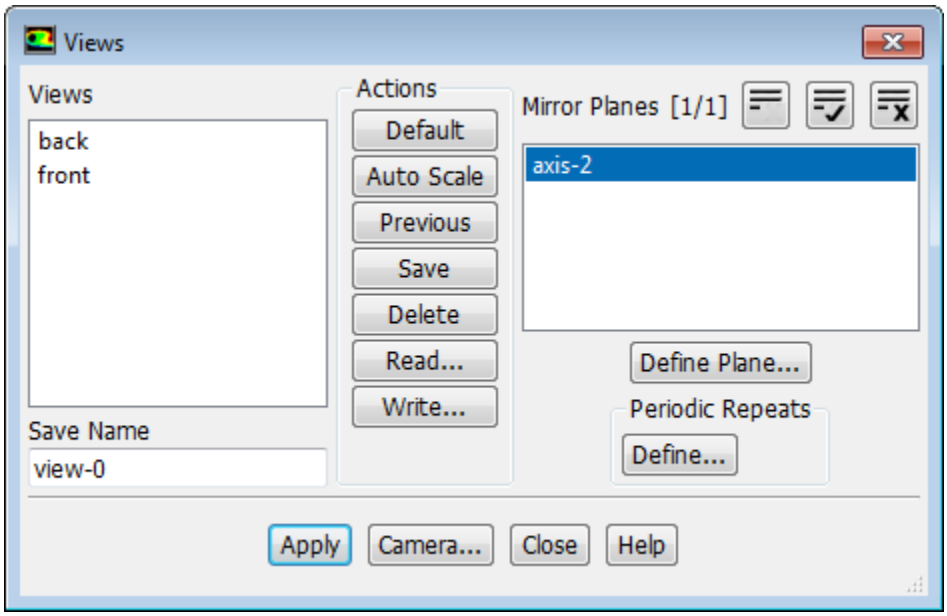
Display
specifies that the frame should encompass the portion of the domain that is shown in the display.

Axes
contains controls for specifying the frame boundaries and measurements. See Adding a Bounding Frame (p. 1823) for instructions on using these items.

Display
updates the active graphics window with the current frame settings.

37.19.24. Views Dialog Box

With the **Views** dialog box, you can make various modifications to the view displayed in the active graphics window. See Modifying the View (p. 1803) for details about the items below.



Controls

Views
lists the currently defined views. Clicking on a view name highlights that name and enters it into the **Name** field. Double-clicking on a view name restores that view in the active graphics window.

Save Name
specifies the name to use when saving a view.

Actions
contains buttons for performing various actions related to the **Views** list and the **Save Name**.

Default
restores the "front" view in the active graphics window.

Auto Scale
modifies the view in the active graphics window by scaling and centering the current scene without changing its orientation.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Previous
allows you to return to previous displays.

Save
stores the view in the active graphics window with the name in the **Save Name** box. See Saving Views (p. 1810) for details.

Delete
removes the selected view name from the **Views** list.

Important

Be careful not to delete any of the pre-defined views.

Read...
opens The Select File Dialog Box (p. 19), in which you can specify the name of a view file to be read. See Reading View Files (p. 1810) for details.

Write...
opens the Write Views Dialog Box (p. 2584), in which you can select the views to be saved to a view file. See Saving Views (p. 1810) for details.

Mirror Planes
displays a list of all symmetry planes in the domain. Mirror images are drawn for all selected symmetry planes. See Mirroring and Periodic Repeats (p. 1811) for details.

Define Plane...
opens the Mirror Planes Dialog Box (p. 2585), in which you can define a mirror plane for a non-symmetric domain.

Periodic Repeats
indicates the number of periodic repetitions to be displayed. See Mirroring and Periodic Repeats (p. 1811) for details.

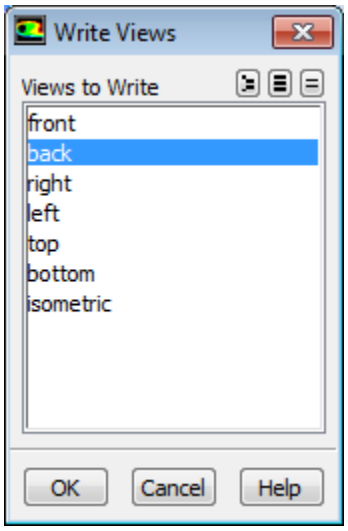
Define...
opens the Graphics Periodicity Dialog Box (p. 2586), in which you can define periodicity for a periodic domain.

Camera...
opens the Camera Parameters Dialog Box (p. 2587).

37.19.25. Write Views Dialog Box

The **Write Views** dialog box allows you to save selected views to a view file. To open it, click **Write...** in the Views Dialog Box (p. 2583). This feature allows you to transfer views between case files. See Saving Views (p. 1810) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

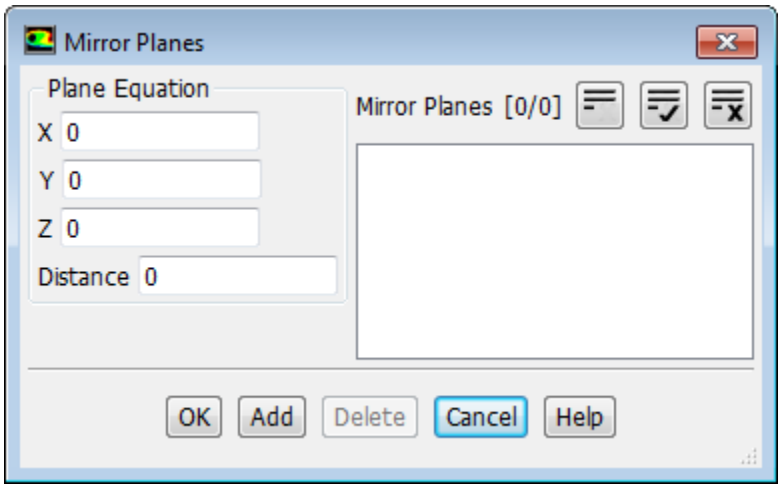


Controls

Views to Write
is a selectable list of the defined views. The selected views will be saved to a view file when you click **OK**.

37.19.26. Mirror Planes Dialog Box

The **Mirror Planes** dialog box allows you to define a symmetry plane for a non-symmetric domain for use with graphics. To open it, click **Define Plane...** in the Views Dialog Box (p. 2583). See Mirroring for Graphics (p. 1815) for details.



Controls

Plane Equation
contains inputs for specifying the equation for the mirror plane: A **X** + B **Y**+ C **Z** = **Distance**.

Mirror Planes
contains a list of all mirror planes you have defined using this dialog box. (Mirror planes that exist in the domain due to symmetry will not appear in this list, since they cannot be modified.)

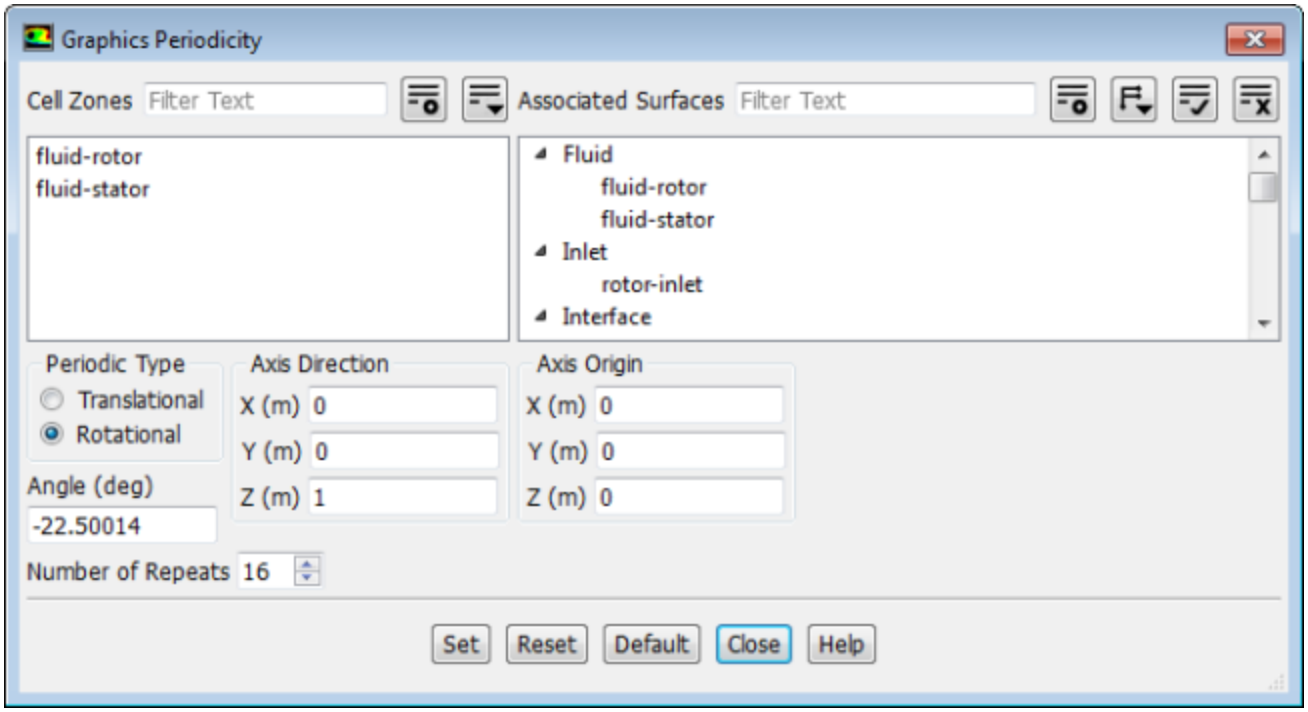
Add
adds the plane defined by the **Plane Equation** to the **Mirror Planes** list.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Delete
deletes the plane(s) selected in the **Mirror Planes** list.

37.19.27. Graphics Periodicity Dialog Box

The **Graphics Periodicity** dialog box allows you to define a periodic repeats, periodic rotation or translation for a non-periodic domain for use with graphics. To open it, click **Define...** under **Periodic Repeats** in the Views Dialog Box (p. 2583). See Periodic Repeats for Graphics (p. 1813) for details.



Controls

Cell Zones
contains the list of the zones in the mesh. You can select one or more zones in this list and specify different periodicity parameters for each zone separately.

Associated Surfaces
contains the list of the surfaces associated with the selected cell zone.

Periodic Type
specifies **Rotational** or **Translational** periodicity.

Angle
specifies the angle by which the domain is rotated to create the periodic repeat. This item is available when you select **Rotational** as the **Periodic Type**.

Translation
specifies the distance in the **X**, **Y**, and **Z** directions by which the domain is translated to create the periodic repeat. This item will appear when you select **Translational** as the **Periodic Type**.

Axis Direction
specifies the direction vector (**X,Y,Z**) for the axis of rotation. This item is available when you select **Rotational** as the **Periodic Type** and the domain is three-dimensional.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Axis Origin
specifies the origin of the axis of rotation. This item is available when you select **Rotational** as the **Periodic Type**.

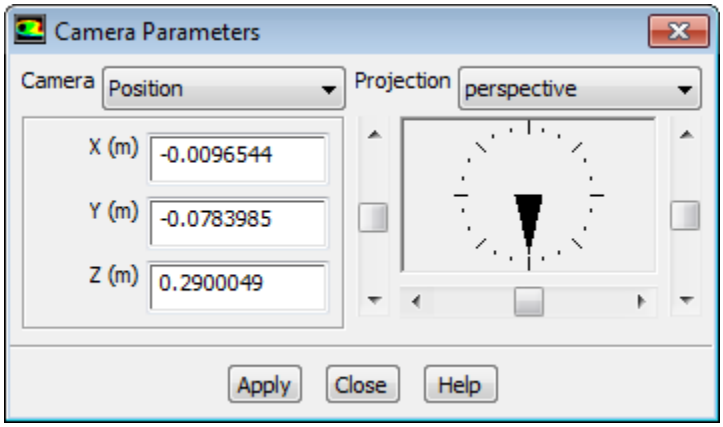
Number of Repeats
specifies the number of times you want to repeat the periodic domain.

Set
applies the periodicity you have defined for the case setup.

Reset
removes any periodicity you have defined, and returns to the default periodicity for the domain (no periodicity for a non-periodic domain).

37.19.28. Camera Parameters Dialog Box

The **Camera Parameters** dialog box allows you to modify the "camera" through which you are viewing the graphics display. See Controlling Perspective and Camera Parameters (p. 1808) for details about the items below.



Controls

Camera
contains a drop-down list of the parameters that define the camera (**Position**, **Target**, **Up Vector**, and **Field**) and **X**, **Y**, and **Z** fields in which you can define the coordinates or field distances for the parameter selected in the drop-down list. Figure 31.43: Camera Definition (p. 1809) illustrates the definition of the camera by these parameters.

Projection
contains a drop-down list that allows you to select a **Perspective** or **Orthographic** view.

(Dial and Sliders)
allow you to rotate and scale the graphics display. The slider on the scale to the left of the dial rotates the display about the horizontal axis at the center of the screen, the slider on the scale below the dial rotates the display about the vertical axis at the center of the screen, and the dial controls rotation about the axis

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

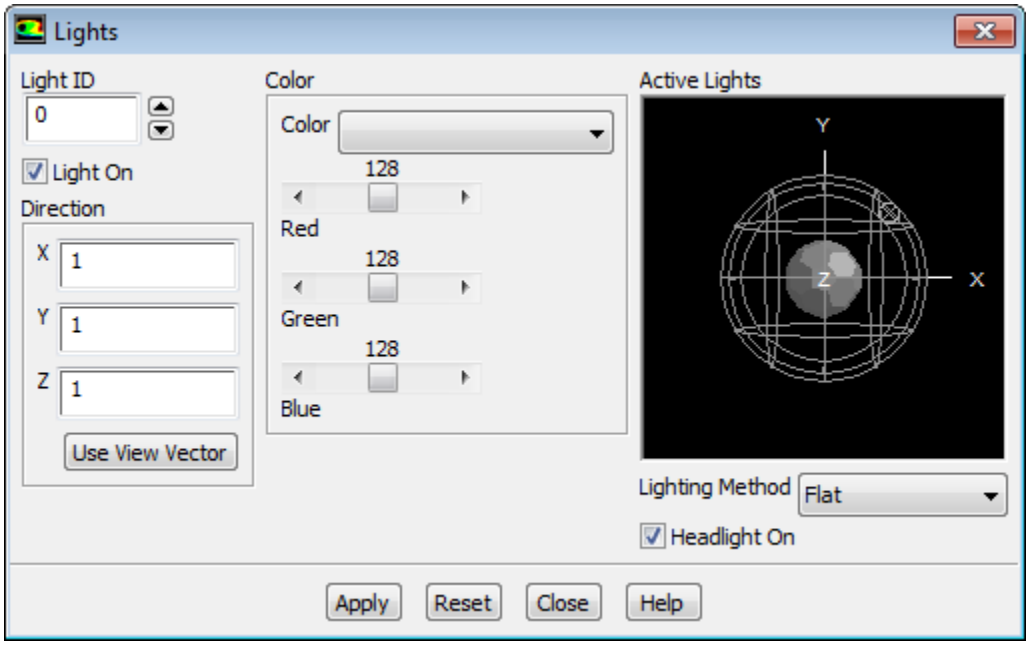
at the center of and perpendicular to the screen. The slider on the scale to the right of the dial zooms in or out in the display. See Rotating the Display (p. 1805) and Zooming the Display (p. 1807) for details.

Important

When you are using the sliders and dial to manipulate the view, you may want to turn off **Wireframe Animation** in the Display Options Dialog Box (p. 2573), so that you can watch the display move interactively while you are moving the slider or the dial indicator.

37.19.29. Lights Dialog Box

The **Lights** dialog box provides an interactive mechanism for placing colored, directional lights in a scene. See Adding Lights (p. 1796) for details about the items below.



Controls

Light ID
indicates the light that is being added, deleted, or modified. By default, light 1 is defined to be dark gray with a direction of (1,1,1).

Light On
indicates whether or not the light specified in **Light ID** is on or off. By turning off the **Light On** option for a particular light, you can remove this light from the display, while still retaining its definition. To add it to the display again, simply turn on the **Light On** button.

Direction
allows you to specify the direction of the light (the position on the unit sphere from which the light emanates) by entering the **X**, **Y**, and **Z** coordinates or by computing the coordinates based on the current view in the graphics window.

X,Y,Z
specify the direction of the light. For example, the direction (1,1,1) means that the rays from the light will be parallel to the vector from (1,1,1) to the origin.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use View Vector

updates the **X,Y,Z** fields with the appropriate values for the current view in the active graphics window, and shines a light in that direction. Instead of entering the **X,Y,Z** values for a light's direction vector, you can use your mouse to change the view in the graphics window so that your position in reference to the geometry is the position from which you would like a light to shine. You can then click the **Use View Vector** button to update the **X,Y,Z** fields with the appropriate values for your current position and update the graphics display with the new light direction. This method is convenient if you know where you want a light to be, but you are not sure of the exact direction vector.

Color

allows you to specify the light color with sliders. You can create your desired color by increasing and decreasing the slider values for the colors **Red**, **Green**, and **Blue**. You can also enter a descriptive string (for example, lavender) in the **Color** field.

Active Lights

shows the position and color of all defined directional lights, and allows you to change the position of a light. All directional lights in ANSYS Fluent are assumed to be at infinity and pass through the unit sphere at the position shown. All light rays arriving at the scene from one light are parallel. The colored markers on the surface of the sphere represent the color and direction of these *distant* lights. These lights point towards the center of the sphere (the origin, which is usually where the geometry is).

Lighting Method

specifies the method to be used in lighting interpolation: **Off**, **Flat**, **Gouraud**, or **Phong**. (**Flat** is the most basic method: there is no interpolation within the individual polygonal facets. **Gouraud** and **Phong** have smoother gradations of color because they interpolate on each facet.) When **Off** is selected, lighting effects are disabled.

Headlight On

enables constant lighting effects in the direction of the view.

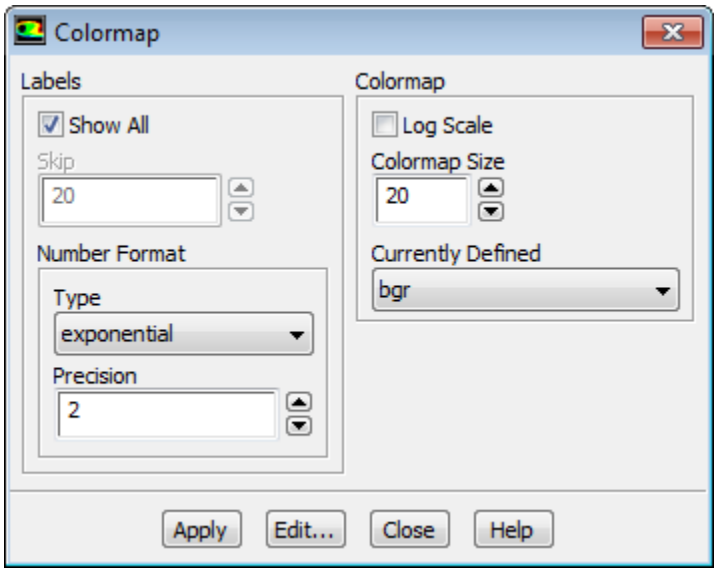
Reset

resets the light definitions to their last saved state (the lighting that was in effect the last time you opened the dialog box or clicked on **Apply**).

37.19.30. Colormap Dialog Box

The **Colormap** dialog box allows you to select and modify existing colormaps. See Selecting a Colormap (p. 1793) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Labels
allows you to customize the display of your colormap labels.

Show All
Enable this option if you want all the labels to show alongside the colormap. Disable if you want to skip some of the label displays.

Skip
sets the number of labels to be skipped.

Number Format
contains controls for changing the format of the labels on the color scale. These labels are the character strings used to define the color divisions at the left of the graphics window.

Type
sets the form of the labels. You may select from a drop-down list of options, including the following:

general
displays the real value with either float or exponential form based on the size of the number and the defined **Precision**.

float
displays the real value with an integral and fractional part (for example, 1.0000), where the number of digits in the fractional part is determined by **Precision**.

exponential
displays the real value with a mantissa and exponent (for example, 1.0e-02), where the number of digits in the fractional part of the mantissa is determined by **Precision**.

Precision
defines the number of fractional digits displayed in the labels.

Colormap
contains controls for the colormap size and scale, and for selection of a defined colormap.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Log Scale
enables the use of a logarithmic scale for the color scale (rather than the default decimal scale). See Specifying the Colormap Size and Scale (p. 1793) for details.

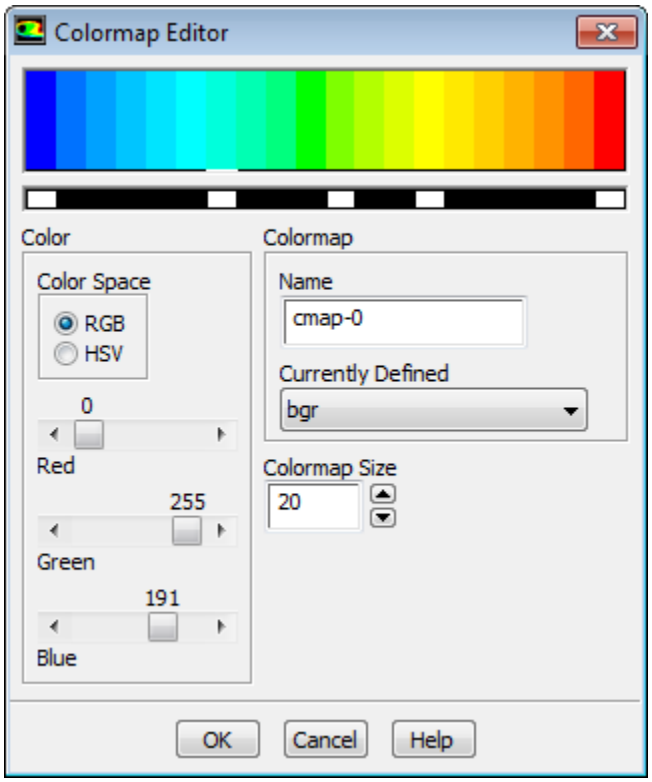
Colormap Size
specifies the number of distinct colors in the colormap. You may specify from 2 to 100 colors.

Currently Defined
contains a drop-down list of all pre-defined colormaps and all custom colormaps defined by you. Select the colormap to be used from this list.

Edit...
opens the Colormap Editor Dialog Box (p. 2591), in which you can create a custom colormap.

37.19.31. Colormap Editor Dialog Box

The **Colormap Editor** dialog box allows you to create custom colormaps. See Creating a Customized Colormap (p. 1795) for details about the items below.



Controls

(The Drawing Area)
is used to interactively modify colormaps. You can use the notion of anchor points to interpolate linearly between two defined anchor points. The colors at the top of the dialog box allow you to preview the colormap that is being defined. The black bar and white squares below the colors allow you to set, delete, and modify anchor points.

Color
specifies color components for the currently selected anchor point.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color Space
gives you a choice of selecting the color specification.

RGB
enables the specification of colors based on their red, green, and blue components.

HSV
enables the specification of colors based on their hue, saturation, and value.

Red, Green, Blue
are scales with which you can specify the RGB components of the selected anchor color. These scales will appear when **RGB** is selected.

Hue, Saturation, Value
are scales with which you can specify the HSV components of the selected anchor color. These scales will appear when **HSV** is selected.

Colormap
lists colormap names and the name of the colormap being edited.

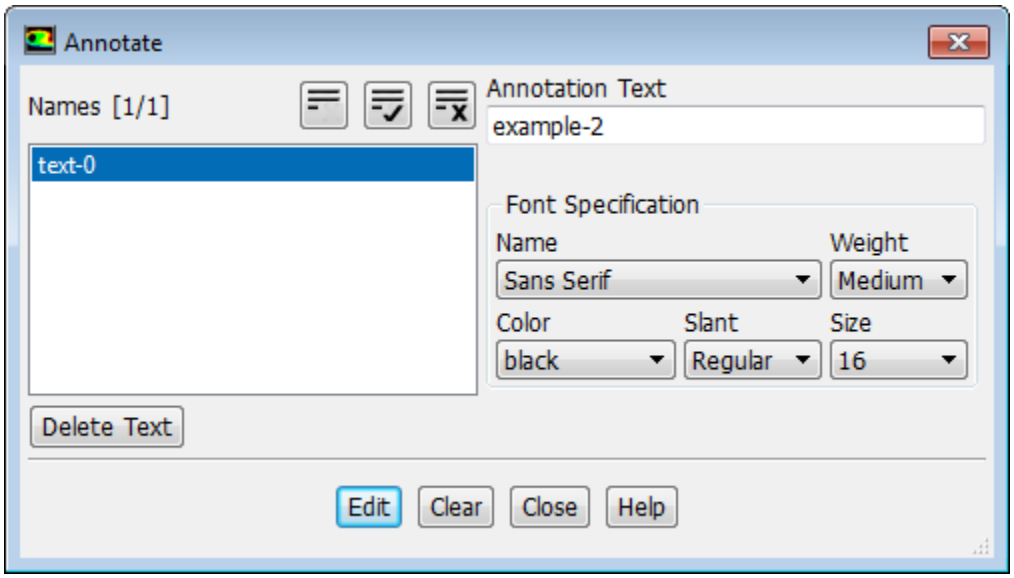
Name
sets the name of the colormap being edited/selected. You can rename existing colormaps and provide names for new colormaps.

Currently Defined
contains a drop-down list of all pre-defined colormaps and all custom colormaps defined by you. You can select a colormap to be modified from this list.

Colormap Size
specifies the number of distinct colors in the colormap. You may specify from 2 to 100 colors.

37.19.32. Annotate Dialog Box

You can use the **Annotate** dialog box to add text with optional attachment lines to the graphics windows, or to modify existing text. See Adding Text Using the Annotate Dialog Box (p. 1790) for details about the items below.



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Names
contains a selectable list of all annotation text strings that have been defined. You can choose a text string to be deleted or edited.

Delete Text
deletes the text strings selected in the **Names** list from the display.

Annotation Text
contains the annotation text string you want to add, or the annotation text string for the item selected in the **Names** list.

Font Specification
contains controls for defining or modifying the font in the annotation text string.

Name
contains a drop-down list of various font styles.

Weight
contains a drop-down list from which you can select **Medium** or **Bold**.

Color
contains a drop-down list of colors that can be used for the text.

Slant
contains a drop-down list from which you can select **Regular** or **Italic** as the slant type.

Size
contains a drop-down list of font sizes (in points).

Add
adds the current **Annotation Text** to the active graphics window. A dialog box will prompt you to select a screen location using the mouse-probe button on your mouse (see Controlling the Mouse Button Functions (p. 1800) for more information on setting the mouse buttons).

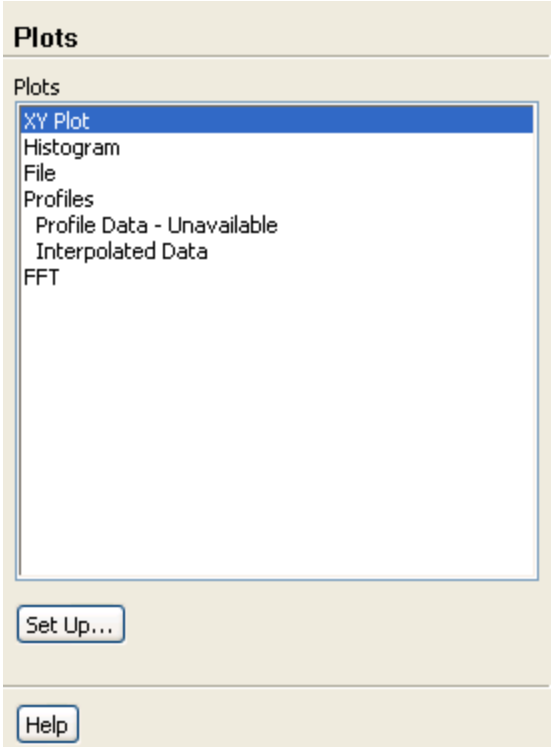
Edit
updates the edited text in the active graphics window. This button will replace the **Add** button when you are editing an existing text string from the **Names** list.

Clear
removes all annotation text and attachment lines from the active graphics window.

37.20. Plots Task Page

The **Plots** task page allows you to create plots (XY, histograms, profiles, and so on) of your computational results. See Displaying Graphics (p. 1747) for more information.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Plots
contains a listing of the various plot types available in ANSYS Fluent.

You can double-click an item in the **Plots** list to open the corresponding dialog box, or you can select the item in the list and click the **Set Up...** button.

XY Plot
selecting this item and clicking the **Set Up...** button opens the Solution XY Plot Dialog Box (p. 2595).

Histogram
selecting this item and clicking the **Set Up...** button opens the Histogram Dialog Box (p. 2598).

File
selecting this item and clicking the **Set Up...** button opens the File XY Plot Dialog Box (p. 2599).

Profiles
two types of profile plots are available:

Profile Data
selecting this item and clicking the **Set Up...** button opens the Plot Profile Data Dialog Box (p. 2601).

Interpolated Data
selecting this item and clicking the **Set Up...** button opens the Plot Interpolated Data Dialog Box (p. 2601).

FFT
selecting this item and clicking the **Set Up...** button opens the Fourier Transform Dialog Box (p. 2602).

Set Up...
displays the dialog box corresponding to the selected item in the **Plots** list.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For additional information, see the following sections:

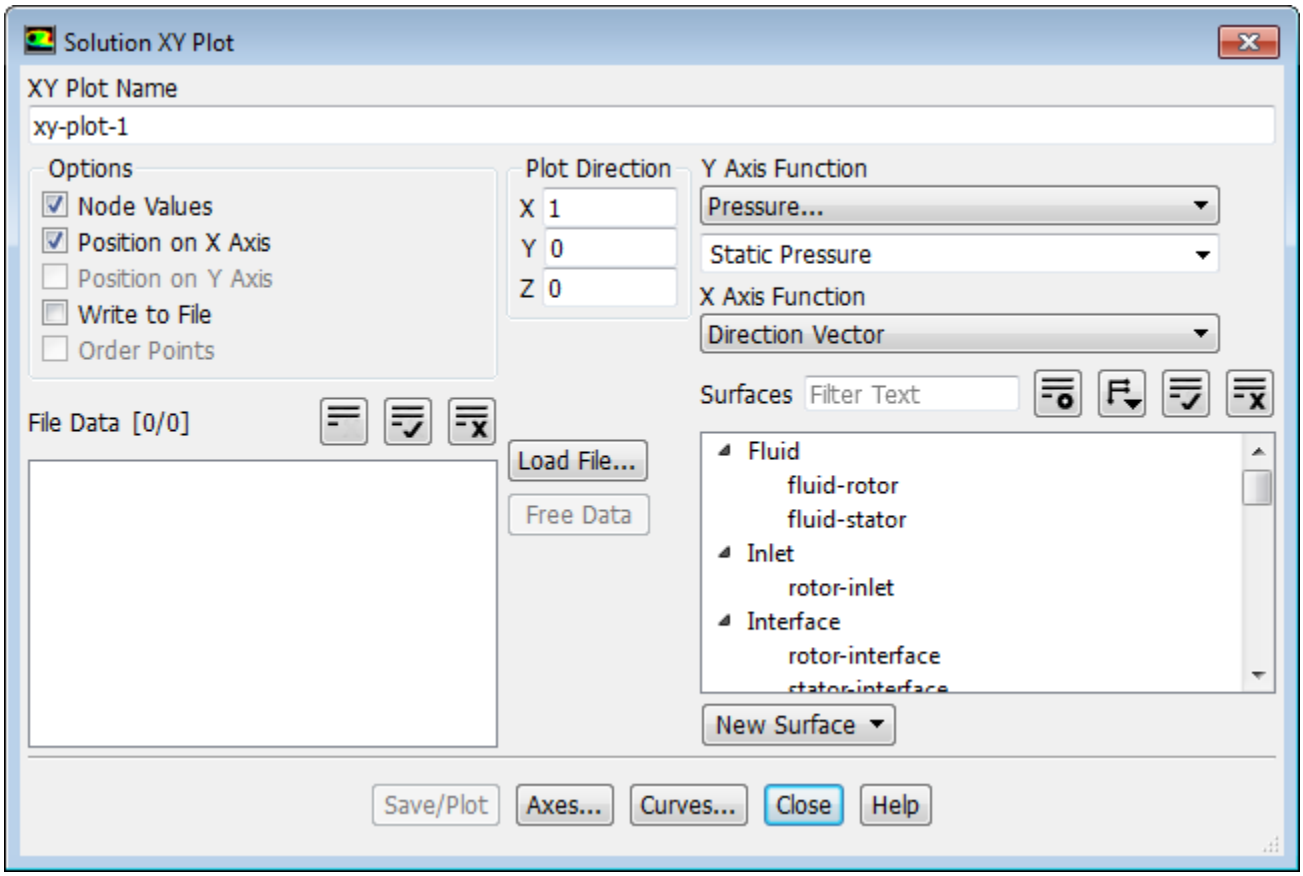
37.20.1. Solution XY Plot Dialog Box
37.20.2. Histogram Dialog Box
37.20.3. File XY Plot Dialog Box
37.20.4. Plot Profile Data Dialog Box
37.20.5. Plot Interpolated Data Dialog Box
37.20.6. Fourier Transform Dialog Box
37.20.7. Plot/Modify Input Signal Dialog Box
37.20.8. Axes Dialog Box
37.20.9. Curves Dialog Box

37.20.1. Solution XY Plot Dialog Box

The **Solution XY Plot** dialog box allows you to display zone, surface and file data in an XY plot format. See Steps for Generating Solution XY Plots (p. 1840) for details about the items below.

*You can also access the **Solution XY Plot** dialog box through the **Postprocessing** ribbon tab.*

Postprocessing → Plots → XY Plot → New...



Controls

XY Plot Name

is the name for a solution XY plot definition. You can specify a name or use the default name **xy-plot-*id***. This control appears only for solution XY plot definitions.

Options

contains check buttons that control the presentation of node or cell-averaged values, the selection of axis functions, and the ability to write the plot data to a file.

Node Values

toggles the node averaging of the data presented in the plot. If the option is inactive, cell values are presented. See Choosing Node or Cell Values (p. 1844) for details.

Position on X Axis, Position on Y Axis

set the x-axis or y-axis function to be the position. If one of these options is turned on, the other will automatically be turned off. You can turn both options off to generate a plot of one flow-field function vs. another, selecting the function for each axis using the **X Axis Function** and **Y Axis Function** drop-down lists.

Write to File

activates the file-writing option. When this option is selected, the **Plot** push button will change to **Write...**. Clicking on the **Write...** button will open the **Select File** dialog box (The Select File Dialog Box (p. 19) and The Select File Dialog Box (Windows) (p. 19)), in which you can specify a name and save a file containing the plot data. The format of this file is described in XY Plot File Format (p. 1850).

Order Points

specifies that plot data being saved to a file should be sorted in order of ascending x axis value. This option is available only when the **Write to File** option is turned on.

Plot Direction

contains inputs for defining the plot direction.

If **Direction Vector** (the default) is selected in the **X Axis Function** or **Y Axis Function** drop-down list (whichever is the position axis), the inputs are the components of the direction vector. The position axis of the plot will have coordinate values that correspond to the dot product of the data coordinate vector with the plot direction vector. See Steps for Generating Solution XY Plots (p. 1840) for details.

X

is the component in the x direction.

Y

is the component in the y direction.

Z

is the component in the z direction.

If **Curve Length** is selected in the **X Axis Function** or **Y Axis Function** drop-down list (whichever is the position axis), the inputs are the direction along the length of the surface selected in the **Surfaces** list. See Steps for Generating Solution XY Plots (p. 1840) for details.

Default

specifies the plot direction as the direction of increasing curve length.

Reverse

specifies the plot direction as the direction of decreasing curve length.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Show
displays the selected surface in the graphics window, marking the start of the surface with a blue dot and the end of the surface with a red dot. ANSYS Fluent will also display arrows on the surface showing the direction in which the variable will be plotted.

Y Axis Function, X Axis Function
contain lists of solution variables that can be used for the y or x axis of the plot. If the **Position on X Axis** option is turned on, **X Axis Function** will become a single drop-down list, containing two options: **Direction Vector** (to plot the selected variable as a function of position along a specified direction vector) and **Curve Length** (to plot the selected variable as a function of position along the length of a specified curvilinear surface). See Steps for Generating Solution XY Plots (p. 1840) for details.

Likewise, if **Position on Y Axis** is turned on, **Y Axis Function** will become a single drop-down list containing **Direction Vector** and **Curve Length**. If both **Position on X Axis** and **Position on Y Axis** are turned off, you can select field functions for both axes using the **X Axis Function** and **Y Axis Function** lists.

File Data
is a selectable list of the plot titles associated with the loaded external data files. You may choose any number of files for the data plot. The files are loaded using the **Load File...** push button. The format of these files is presented in XY Plot File Format (p. 1850). See Including External Data in the Solution XY Plot (p. 1844) for details.

Load File...
opens the **Select File** dialog box (The Select File Dialog Box (p. 19)), in which you can select the plot file to be read. See Including External Data in the Solution XY Plot (p. 1844) for details. After the external file is loaded, its plot title will be displayed in the **File Data** list.

Free Data
removes the files selected in the **File Data** list.

Surfaces
is a selectable list of surfaces in the solution domain. You may choose any number of surfaces for the data plot.

New Surface
is a drop-down list button that contains a list of surface options:

Point
opens the Point Surface Dialog Box (p. 2780).

Line/Rake
opens the Line/Rake Surface Dialog Box (p. 2727).

Plane
opens the Plane Surface Dialog Box (p. 2777).

Quadric
opens the Quadric Surface Dialog Box (p. 2781).

Iso-Surface
opens the Iso-Surface Dialog Box (p. 2721).

Iso-Clip
opens the Iso-Clip Dialog Box (p. 2720).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plot
plots the specified surface and/or file data in the active graphics window using the current axis and curve attributes. If the **Write to File** option is turned on, this button becomes the **Write...** button.

Write...
opens the **Select File** dialog box (The Select File Dialog Box (p. 19)), in which you can save the plot data to a file. This button replaces the **Plot** button when the **Write to File** option is turned on.

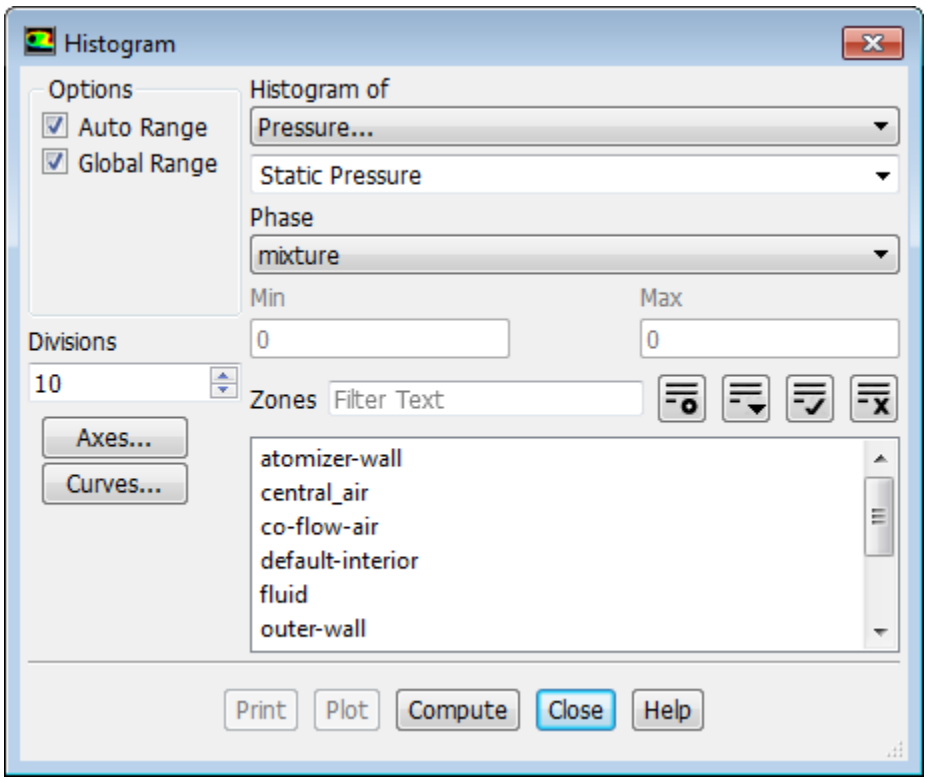
Axes...
opens the Axes Dialog Box (p. 2608), which allows you to customize the plot axes.

Curves...
opens the Curves Dialog Box (p. 2610), which allows you to customize the curves used in the XY plot.

Save/Plot
plots the specified surface and/or file data in the active graphics window using the current axis and curve attributes and saves the XY plot definition. This button appears only for solution XY plot definitions and replaces the **Plot** button.

37.20.2. Histogram Dialog Box

The **Histogram** dialog box allows you to create histograms of selected geometric or physical data. See Steps for Generating Histogram Plots (p. 1851) for details about the items below.



Controls

Options
contains the check buttons for current histogram options.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Auto Range
toggles the ability to specify the minimum and maximum range of scalar values in the histogram print or plot. If the option is not active, the **Min** and **Max** fields are editable, and you may specify the desired range. If the option is active, the range is defined by the minimum and maximum values in the computational domain.

Global Range
toggles the ability to specify the minimum and maximum values on the range of values on the selected surfaces, rather than in the entire domain.

Divisions
sets the number of data intervals that will exist in the histogram.

Zone Types
contains a list of available face or cell zone types.

Axes...
opens the Axes Dialog Box (p. 2608), which allows you to customize the plot axes.

Curves...
opens the Curves Dialog Box (p. 2610), which allows you to customize the curves used in the histogram plot.

Histogram of
contains a list of scalar quantities that can be used in the histogram.

Min
displays or allows definition of the minimum value of the selected scalar quantity used in the histogram.

Max
displays or allows definition of the maximum value of the selected scalar quantity used in the histogram.

Zones
contains a list of available face or cell zones.

Print
displays the histogram interval.

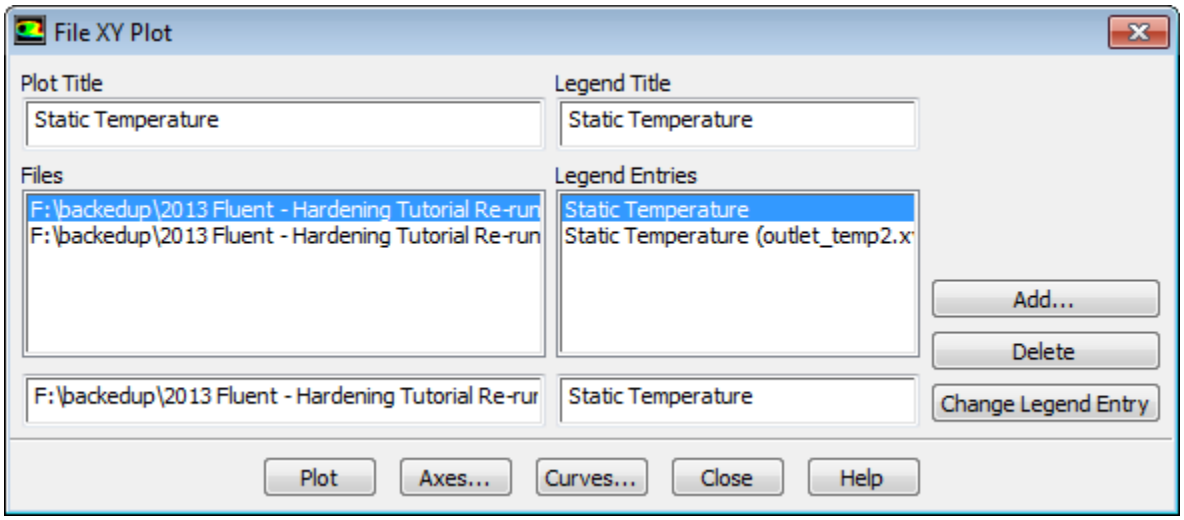
Plot
displays a plot of the percentage of the total number of cells versus the scalar quantity in the active graphics window.

Compute
computes the minimum and maximum cell values of the selected scalar quantity. The values are displayed in **Min** and **Max**.

37.20.3. File XY Plot Dialog Box

The **File XY Plot** dialog box allows you to display data read from one or more files in an abscissa/ordinate plot form. The format of the plot file is described in XY Plot File Format (p. 1850). See Steps for Generating XY Plots of Data in External Files (p. 1844) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Plot Title

displays the current plot title. The default plot title is the y-axis label of the first file read into the dialog box, but you can edit the text entry to change the plot title.

Files

contains a selectable list of loaded file names. If you enter a valid file name into the text entry field below the list and click the **Add...** button ANSYS Fluent will load the file and update the dialog box. If, however, the file is not found or has already been read, The Select File Dialog Box (p. 19) will be opened.

Legend Title

sets the title of the legend. By default, the legend has no title.

Legend Entries

contains a selectable list of legend labels associated with the loaded files. The default legend label is the title (first string) in the file. You can modify the legend label by selecting the old name in the **Legend Entries** list. It is then displayed in the text entry field below the list and may be edited.

Add...

loads the file named in the **Files** text entry field or opens the **Select File** dialog box if no name, a wrong name, or a duplicate name appears in that text field.

Delete

removes the file selected in the **Files** list.

Change Legend Entry

changes the legend label of the selected file to the text entered in the text entry field below the **Legend Entries** list.

Plot

displays an XY plot of the data associated with every loaded file. You can use the **Delete** button to remove files that you do not want to plot.

Axes...

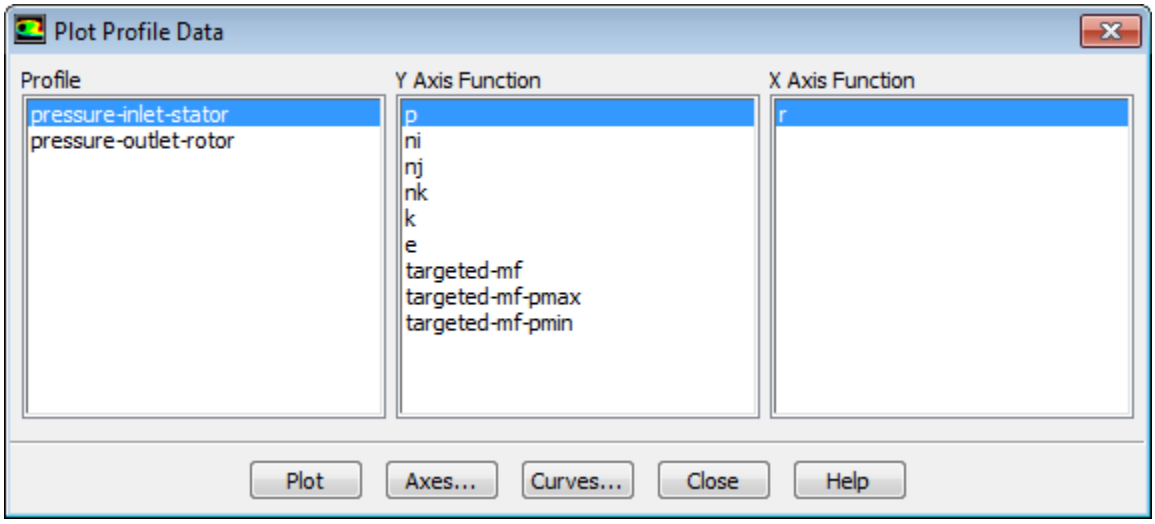
opens the Axes Dialog Box (p. 2608), which allows you to customize the plot axes.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Curves...
opens the Curves Dialog Box (p. 2610), which allows you to customize the curves used in the XY plot.

37.20.4. Plot Profile Data Dialog Box

The **Plot Profile Data** dialog box allows you to display an XY plot of the original data points of a boundary profile before it is interpolated onto the cell faces of a boundary. See Steps for Generating Plots of Profile Data (p. 1846) for details about the items below.



Controls

Profile
contains a selectable list of available profiles. When a profile is selected its available fields are displayed under **Y Axis Function**.

Y Axis Function
contains a selectable list of the fields available in the selected profile that can be used for the y axis of the plot.

X Axis Function
contains a selectable list of the variables that can be used for the x axis of the plot.

Plot
displays an XY plot of the data points from the selected profile.

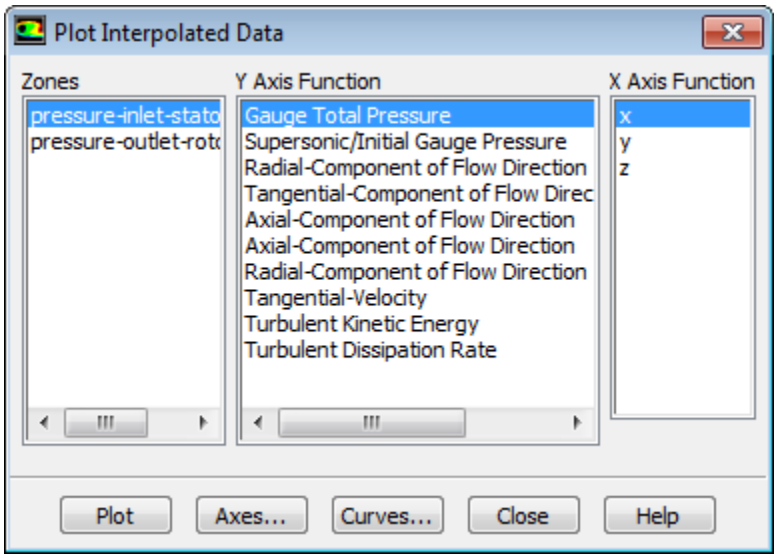
Axes...
opens the Axes Dialog Box (p. 2608), which allows you to customize the plot axes.

Curves...
opens the Curves Dialog Box (p. 2610), which allows you to customize the curves used in the XY plot.

37.20.5. Plot Interpolated Data Dialog Box

The **Plot Interpolated Data** dialog box allows you to display an XY plot of the values assigned to the cell faces when a profile file has been interpolated on a boundary. See Steps for Generating Plots of Interpolated Profile Data (p. 1847) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zones
contains a selectable list of the zones for which a profile field has been set as one or more of the parameters are displayed under **Y Axis Function**.

Y Axis Function
contains a selectable list of the profile-related parameters in the selected zone that can be used for the y axis of the plot. The name of the parameter will be the same as that of the drop-down list in the boundary condition dialog box from which the profile field was selected.

X Axis Function
contains a selectable list of the geometry variables that can be used for the x axis of the plot.

Plot
displays an XY plot of the cell face values (as interpolated from the data points of the profile file) of the selected zone.

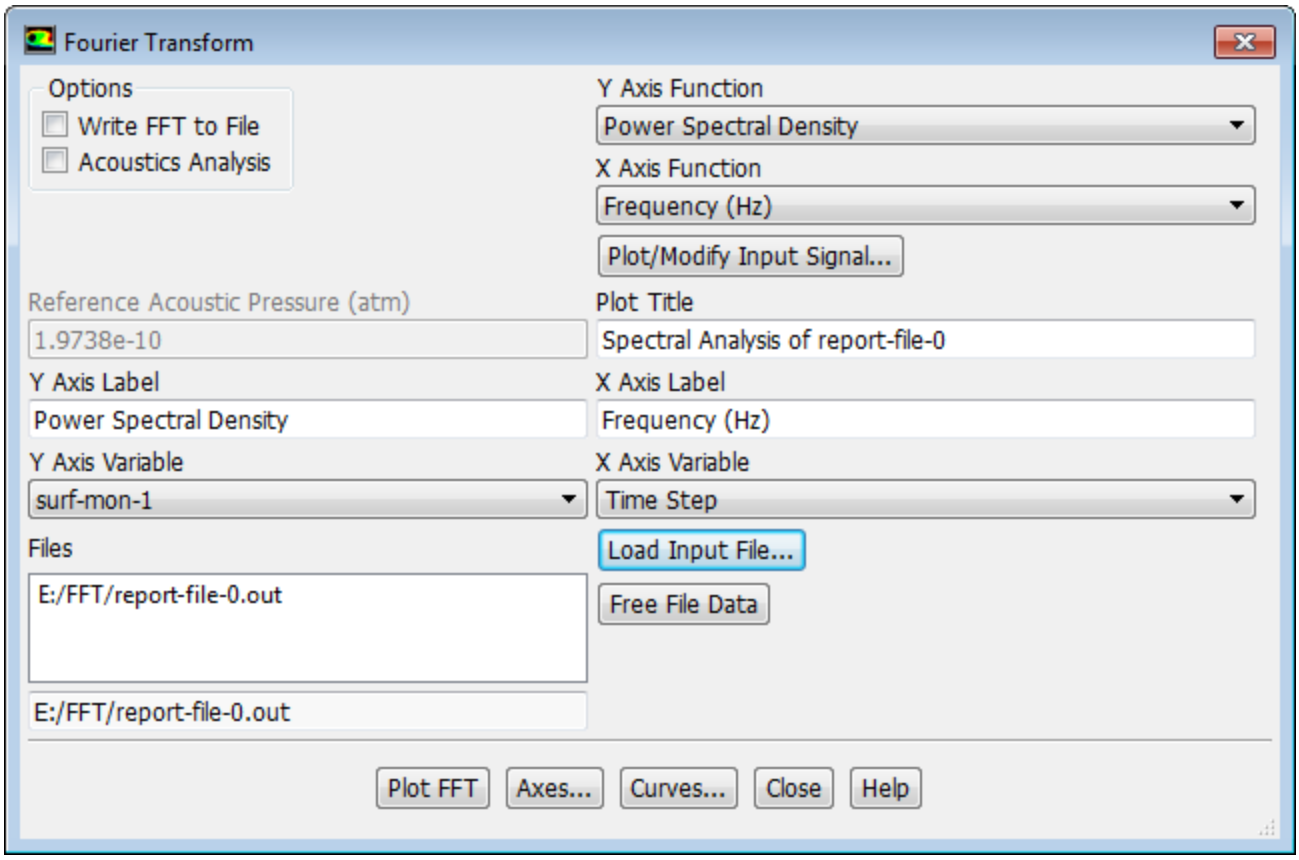
Axes...
opens the Axes Dialog Box (p. 2608), which allows you to customize the plot axes.

Curves...
opens the Curves Dialog Box (p. 2610), which allows you to customize the curves used in the XY plot.

37.20.6. Fourier Transform Dialog Box

The **Fourier Transform** dialog box allows you to analyze your time dependent data using the Fast Fourier Transform (FFT) algorithm. See Fast Fourier Transform (FFT) Postprocessing (p. 1873) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options
lets you write the FFT data to a file or display the FFT data in a graphics window.

Write FFT to File
allows you to write out the FFT data directly to a file. When selected, the **Plot FFT...** buttons becomes the **Write FFT...** button.

Acoustics Analysis
enables the acoustics-relevant spectrum options (sound pressure level spectra and frequency band charts), and specifies that the overall sound pressure level is calculated and displayed in the console when you plot or write the FFT data. All dB-measured quantities are based on the **Reference Acoustic Pressure**.

Process Options
contains options to analyze signal data.

Process Receiver
allows you to analyze receiver data stored in memory.

Process File Data
allows you to analyze signal data from an existing input file.

Y Axis Function
contains a list from which you can select the function for the y axis.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X Axis Function
contains a list from which you can select the function for the x axis.

Receiver
contains a list of receivers from which you can select when the **Process Receiver** option is enabled.

Plot/Modify Input Signal...
opens the Plot/Modify Input Signal Dialog Box (p. 2605), which you can use to plot the input signal, as well as customize the data set and define spectrum smoothing in preparation for applying the FFT algorithm.

Reference Acoustic Pressure
allows you to specify the reference acoustic pressure (for example, p_{ref} in Equation 31.41 (p. 1880)) used to calculate the dB-measured quantities when the **Acoustics Analysis** option is enabled.

Plot Title
lets you create a new title or edit the original title for the FFT plot. By default, ANSYS Fluent adds the string "Spectral Analysis of" to the title originally applied to the input signal plot.

Y Axis Label
allows you to create a new y axis label or edit the original y axis label. By default, the **Y Axis Label** corresponds to the selection in the **Y Axis Function** drop-down list.

X Axis Label
allows you to create a new x axis label or edit the original x axis label. By default, the **X Axis Label** corresponds to the selection in the **X Axis Function** drop-down list.

Y Axis Variable
if your input report file contains multiple report definitions, this allows you to specify the y axis variable.

X Axis Variable
if your input report file contains multiple report definitions, this allows you to specify the x axis variable.

Files
lists the loaded input signal data files.

Load Input File...
loads an input signal data file into ANSYS Fluent for FFT analysis. The input file is listed under **Files**.

Free File Data
removes data from FFT analysis once the input signal data file is selected in the **Files** list.

Plot FFT
displays the FFT data in a graphic window and, if **Acoustics Analysis** is enabled, calculates the overall sound pressure level in dB (based on the **Reference Acoustic Pressure**) and displays it in the console. If the **Write FFT to File** option is selected, then this button becomes the **Write FFT** button, which opens a file selection dialog box so that you write the FFT data to a file.

Write FFT
opens The Select File Dialog Box (p. 19) in which you can specify a name and save the FFT data to a file. If **Acoustics Analysis** is enabled, the overall sound pressure level will be calculated in dB (based on the **Reference Acoustic Pressure**) and displayed in the console at this time. If the **Write FFT to File** option is selected, then the **Plot FFT** button changes to the **Write FFT** button.

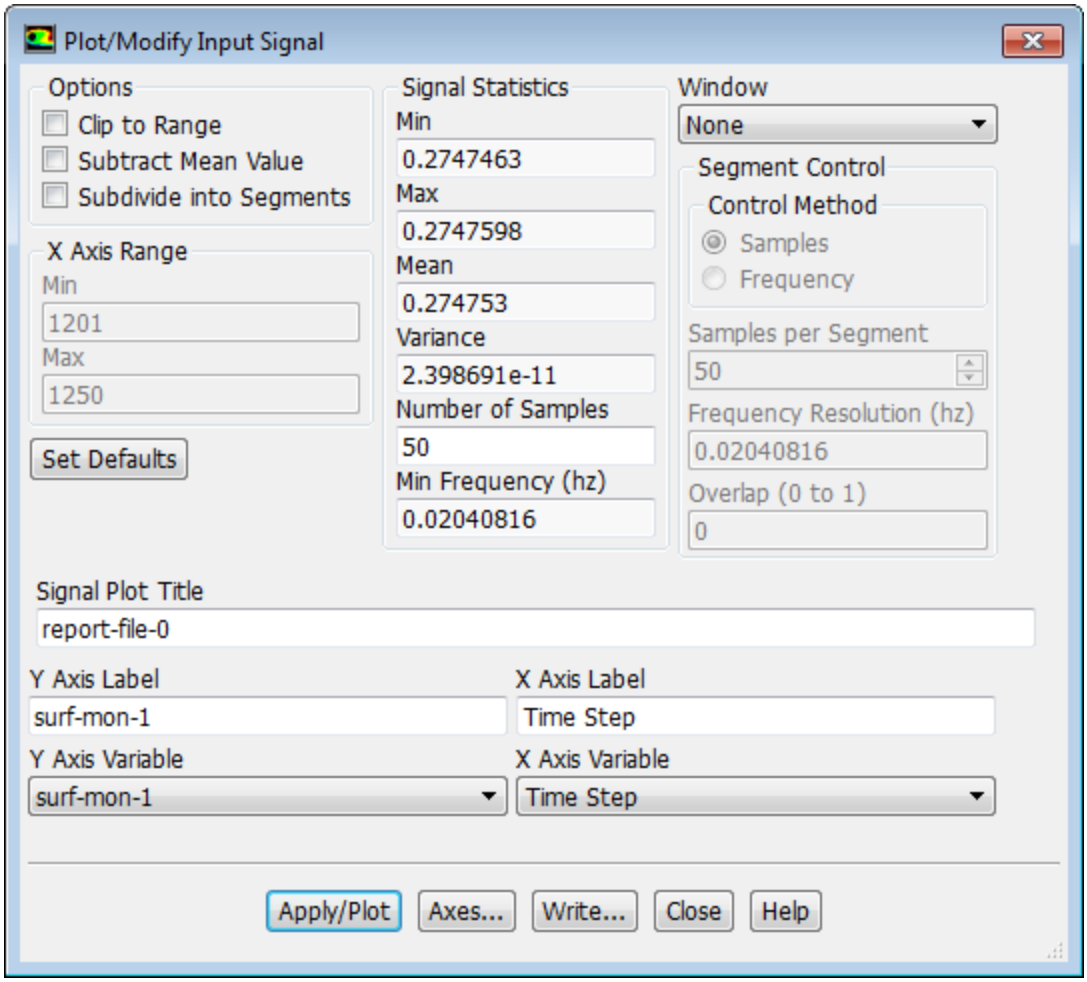
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Axes...
opens the Axes Dialog Box (p. 2608), which allows you to customize the plot axes.

Curves...
opens the Curves Dialog Box (p. 2610), which allows you to customize the curves used in the plot.

37.20.7. Plot/Modify Input Signal Dialog Box

The **Plot/Modify Input Signal** dialog box allows you to plot the input signal, as well as customize the data set and define spectrum smoothing in preparation for applying the FFT algorithm. It is opened from the Fourier Transform Dialog Box (p. 2602). See Using the FFT Utility (p. 1875) for details about the items below.



Controls

Options
allows you to process some or all of the input signal data.

Clip to Range
allows you to analyze a portion of the input signal by specifying data range.

Subtract Mean Value
reduces y axis quantities by the mean value of the relevant signal property.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Subdivide into Segments
enables / disables spectrum smoothing, which can suppress spurious amplitude fluctuations in the Fourier spectrum by splitting the input signal into multiple overlapping segments, so that ANSYS Fluent can apply the FFT algorithm on each segment and then average the resulting spectra. The spectrum smoothing is controlled by the settings in the **Segment Control** group box. See Customizing the Input and Defining the Spectrum Smoothing (p. 1877) for further details.

Signal Statistics
displays signal information.

Min
displays the minimum value for the input signal.

Max
displays the maximum value for the input signal.

Mean
displays the average value for the input signal.

Variance
displays the variance for the input signal.

Number of Samples
displays the total number of samples in the input signal data set.

Min Frequency
displays the finest possible frequency resolution for the input signal.

X Axis Range
allows for a portion of the input signal to be analyzed when the **Process the whole data** option is turned off.

Min
specifies a minimum value for the x axis.

Max
specifies a maximum value for the x axis.

Set Defaults
resets the x axis minimum and maximum values to their original value and turns on the **Process the whole data** option.

Signal Plot Title
lets you create a new title or edit the original title for the input signal plot. By default, the **Signal Plot Title** corresponds with the title originally applied to the input signal plot.

Y Axis Label
allows you to create a new y axis label or edit the original y axis label. By default, the **Y Axis Label** corresponds with the y axis label originally applied to the input signal plot.

X Axis Label
allows you to create a new x axis label or edit the original x axis label. By default, the **X Axis Label** corresponds with the x axis label originally applied to the input signal plot.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Y Axis Variable
if your input report file contains multiple report definitions, this allows you to specify the y axis variable.

X Axis Variable
if your input report file contains multiple report definitions, this allows you to specify the x axis variable.

Window
lets you specify a windowing technique to remove discontinuities in the FFT calculation.

None
applies no windowing technique. This is the default setting.

Hamming
applies the Hamming technique.

Hanning
applies the Hanning technique.

Barlett
applies the Barlett technique.

Blackman
applies the Blackman technique.

Segment Control
provides settings that control the spectrum smoothing when **Subdivide into Segments** is enabled.

Control Method
allows you to specify how you want to define the segment size from the following:

Samples
bases the segment size on the **Samples per Segment**.

Frequency
bases the segment size on the **Frequency Resolution** Δf in Hertz units.

Samples per Segment
defines the number of samples in each segment when **Samples** is selected from the **Control Method** list.

Frequency Resolution
defines the desired frequency resolution Δf in Hertz units when **Frequency** is selected from the **Control Method** list.

Overlap
ranges from 0 to 1 and defines how much adjacent segments overlap. This setting is applicable regardless of which **Control Method** you select.

Apply/Plot
applies any changes you have made in the dialog box and displays the input signal data in a graphics window.

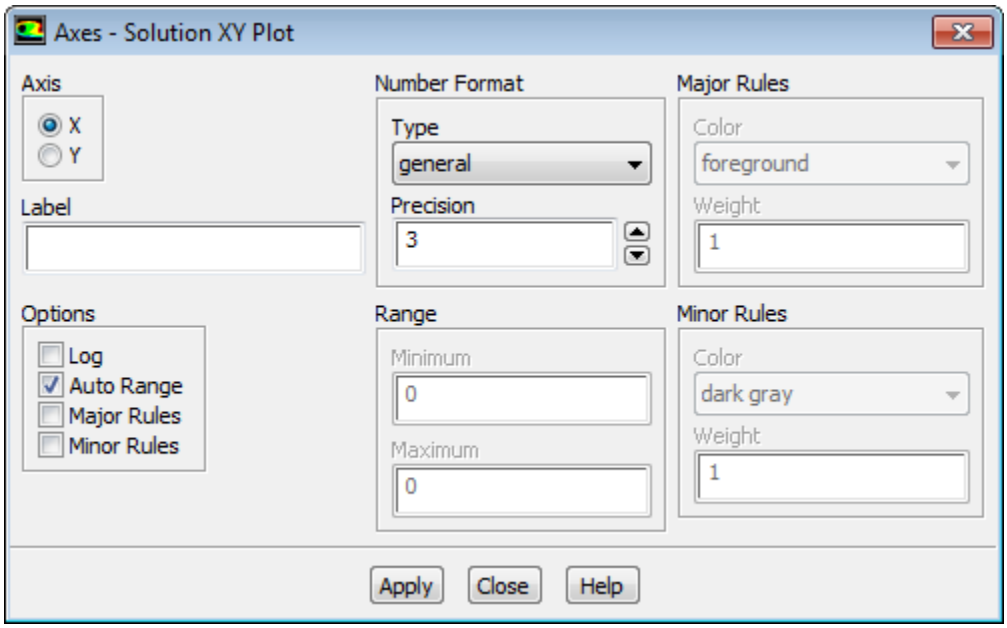
Axes...
opens the Axes Dialog Box (p. 2608), which allows you to customize the plot axes.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Write...
opens The Select File Dialog Box (p. 19), in which you can save the signals to a file.

37.20.8. Axes Dialog Box

The **Axes** dialog box allows you to independently control the characteristics of the ordinate and abscissa on an XY plot or histogram. You can change the labels, scale, range, number format, and major and minor rules visibility and appearance. Note that the title following **Axes** in the dialog box indicates which plot environment you are changing. You can set different parameters for each type of plot that ANSYS Fluent can generate. See Using the Axes Dialog Box (p. 1853) for details about the items below.



Controls

Axis
contains check buttons that allow you to set abscissa (x-axis) or ordinate (y-axis) characteristics.

X
allows you to specify the abscissa characteristics.

Y
allows you to specify the ordinate characteristics.

Label
defines the character string that will label the active axis (the one selected in **Axis**) in the display.

Options
contains check buttons that (de)activate scale, range, major rules, and minor rules.

Log
toggles logarithmic scaling of the active axis. By default, decimal scaling is used.

Auto Range
toggles automatic computation of the range of the active axis. If you deactivate this option, you may input the **Minimum** and **Maximum** values in the **Range** box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Major Rules
toggles the display of major rules on the active axis. Major rules are the horizontal or vertical lines that mark the primary data divisions and span the whole plot window to produce a "grid."

Minor Rules
toggles the display of minor rules on the active axis. Minor rules are the horizontal or vertical lines that mark the secondary data divisions and span the whole plot window to produce a "grid."

Number Format
contains controls for changing the format of the data labels on the active axis. Data labels are the character strings used to define the primary data divisions on the axes.

Type
sets the form of the data labels. You may select from a drop-down list of options, including the following:

general
displays the real value with either float or exponential form based on the size of the number and the defined **Precision**.

float
displays the real value with an integral and fractional part (for example, 1.0000), where the number of digits in the fractional part is determined by **Precision**.

exponential
displays the real value with a mantissa and exponent (for example, 1.0e-02), where the number of digits in the fractional part of the mantissa is determined by **Precision**.

Precision
defines the number of fractional digits displayed in the data labels.

Range
contains the range or extents of the active axis. To set the range manually, you must turn off **Auto Range**. Otherwise the extents are computed automatically.

Minimum
sets the minimum data value for the active axis.

Maximum
sets the maximum data value for the active axis.

Major Rules
contains controls for modifying the appearance of the major rules. To use these controls you must activate **Major Rules** in the **Options** list.

Color
sets the color of the major rules from a drop-down list with numerous color selections.

Weight
sets the line thickness of the major rule. A line of weight 1.0 is normally 1 pixel wide. A weight of 2.0 would make the line twice as thick (2 pixels wide).

Minor Rules
contains controls for modifying the appearance of the minor rules. To use these controls you must activate **Minor Rules** in the **Options** list.

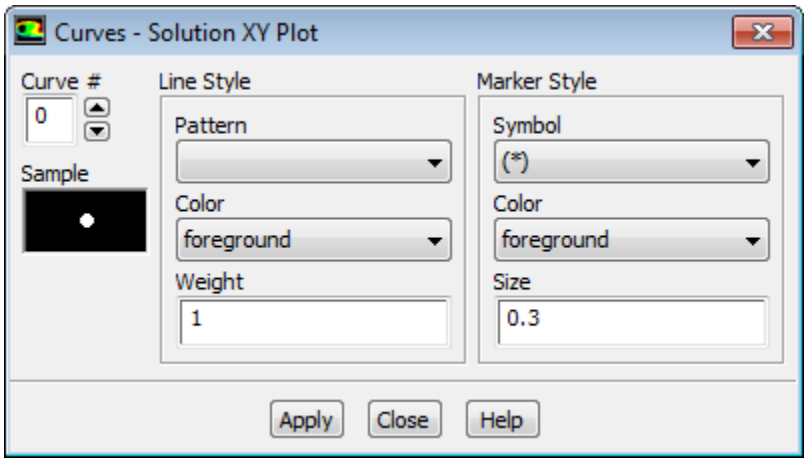
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color
sets the color of the minor rules from a drop-down list with numerous color selections.

Weight
sets the line thickness of the minor rule. A line of weight 1.0 is 1 pixel wide. A weight of 2.0 would make the line twice as thick (2 pixels wide).

37.20.9. Curves Dialog Box

The **Curves** dialog box allows you to modify the appearance of the lines and markers used in XY plots. Note that the title following **Curves** in the dialog box indicates which plot environment you are changing. You can set different parameters for each type of plot that ANSYS Fluent can generate. See Using the Curves Dialog Box (p. 1855) for details about the items below.



Controls

Curve #
defines the active curve number. The present and future marker and line styles apply to the defined curve number. The curves are numbered sequentially, starting from 0. For example, if you were plotting flow-field values on two zones, the first zone would be curve 0, and the second, curve 1. If the plot contains only one curve, the **Curve #** is set to 0 and is not editable.

Sample
displays a single marker and line with the current style attributes.

Line Style
contains controls for modifying the appearance of the active curve.

Pattern
sets the pattern of the active curve. A drop-down list allows you to set the line pattern. Except for **center** and **phantom** lines, the list displays examples of the pattern choices. A **center** line alternates a very long dash and a short dash and a **phantom** line alternates a very long dash and a double short dash.

Color
sets the color of the active curve. A drop-down list allows you to select from a list of color names.

Weight
sets the thickness of the active curve. A line weight of 1.0 is normally 1 pixel wide. Therefore, a weight of 2.0 would make the line twice as thick (2 pixels wide).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Marker Style
contains controls for modifying the appearance of the active curve's marker.

Symbol
sets the symbol used to mark data. You can select the symbol from a drop-down list that contains all the symbol choices. The **Sample** box will allow you to experiment with various markers. For example, in plotting pressure-coefficient data on the upper and lower surfaces of an airfoil, the symbol /*\ (filled-in upward-pointing triangle) could be used for the marker representing the upper surface data, and the symbol */ (filled-in downward-pointing triangle) could be used for the marker representing the lower surface data.

Color
sets the color of the marker on the active line number. A drop-down list allows you to select from a list of color names.

Size
sets the size of the data marker. A symbol of size 1.0 is 3.0% of the height of the display screen, except for the "." symbol, which is always one pixel.

37.21. Reports Task Page

The **Reports** task page allows you to set up reports for your CFD simulation. Reports can be compiled for fluxes, forces, projected areas, surface and volume integrals, among others. See Reporting Alphanumeric Data (p. 1885) for more information.

Controls

Reports
displays a list of available report types in ANSYS Fluent.

You can double-click an item in the **Reports** list to open the corresponding dialog box, or you can select the item in the list and click the **Set Up...** button.

Fluxes
selecting this item and clicking the **Set Up...** button opens the Flux Reports Dialog Box (p. 2613).

Forces
selecting this item and clicking the **Set Up...** button opens the Force Reports Dialog Box (p. 2614).

Projected Areas
selecting this item and clicking the **Set Up...** button opens the Projected Surface Areas Dialog Box (p. 2615).

Surface Integrals
selecting this item and clicking the **Set Up...** button opens the Surface Integrals Dialog Box (p. 2616).

Volume Integrals
selecting this item and clicking the **Set Up...** button opens the Volume Integrals Dialog Box (p. 2620).

Discrete Phase
allows you to report on one of the following three report types:

Sample
selecting this item and clicking the **Set Up...** button opens the Sample Trajectories Dialog Box (p. 2622).

Histogram
selecting this item and clicking the **Set Up...** button opens the Trajectory Sample Histograms Dialog Box (p. 2623).

Summary
selecting this item and clicking the **Set Up...** button opens the Particle Summary Dialog Box (p. 2625).

Heat Exchanger
selecting this item and clicking the **Set Up...** button opens the Heat Exchanger Report Dialog Box (p. 2625).

Set Up...
opens the dialog box corresponding to the selected item in the **Reports** list.

Parameters...
opens the Parameters Dialog Box (p. 2627).

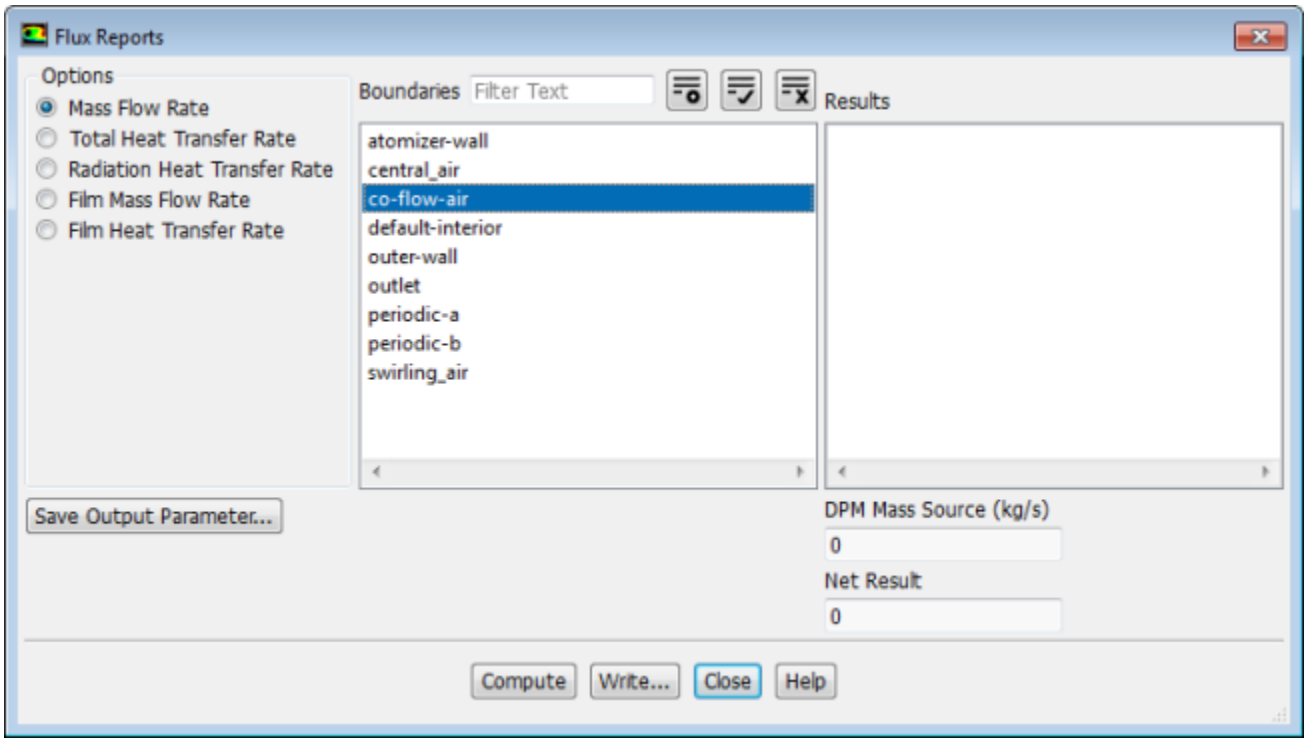
For additional information, see the following sections:
37.21.1. Flux Reports Dialog Box
37.21.2. Force Reports Dialog Box
37.21.3. Projected Surface Areas Dialog Box
37.21.4. Surface Integrals Dialog Box
37.21.5. Volume Integrals Dialog Box
37.21.6. Sample Trajectories Dialog Box
37.21.7. Trajectory Sample Histograms Dialog Box
37.21.8. Particle Summary Dialog Box
37.21.9. Heat Exchanger Report Dialog Box
37.21.10. Parameters Dialog Box
37.21.11. Use Input Parameter in Scheme Procedure Dialog Box
37.21.12. Use Input Parameter for UDF Dialog Box
37.21.13. Rename Dialog Box
37.21.14. Input Parameter Properties Dialog Box

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.21.15. Save Output Parameter Dialog Box

37.21.1. Flux Reports Dialog Box

The **Flux Reports** dialog box allows you to compute the mass flow rate, heat transfer rate, and radiation heat transfer rate on selected boundary zones. See Fluxes Through Boundaries (p. 1911) for details.



Controls

Options
contains the following option buttons:

Mass Flow Rate
turns on the computation of the mass flow rate for the selected boundary zones.

Total Heat Transfer Rate
turns on the computation of the total heat transfer rate for the selected boundary zones.

Total Sensible Heat Transfer Rate
turns on the computation of the total sensible heat transfer rate for the selected boundary zones. It reports the total energy flux as defined in Equation 5.2 in the Theory Guide.

Radiation Heat Transfer Rate
turns on the computation of the radiation heat transfer rate for the selected boundary zones.

Film Mass Flow Rate
(available only when the Eulerian wall film model is enabled) turns on the computation of the mass flow rate for the selected boundary zone(s).

Film Heat Transfer Rate
(available only when the Eulerian wall film model is enabled) turns on the computation of the film heat transfer rate for the selected boundary zone(s).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Save Output Parameter...
opens the Save Output Parameter Dialog Box (p. 2631).

Boundaries
contains a selectable list of valid boundary zones for flux reporting.

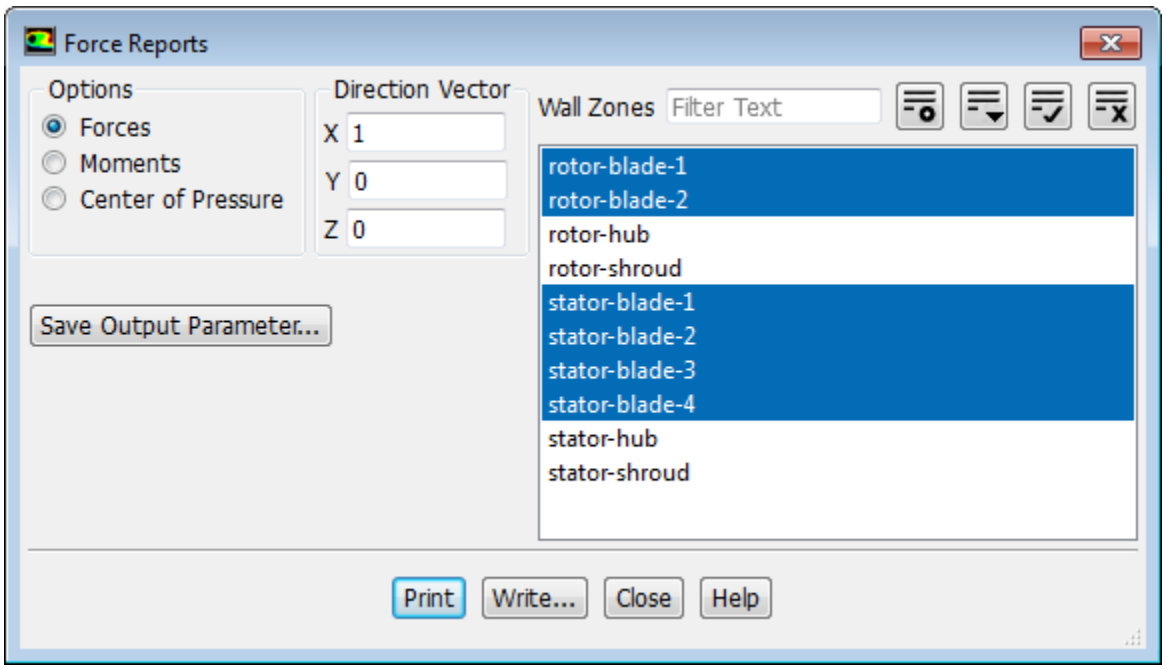
Results
displays the results of the selected flux computation for each boundary zone selected. The summation of the individual zone flux results is displayed in the box below the **Results** list.

Compute
computes the flux for each of the selected boundary zones and updates the **Heat of Reaction Source** and **Net Results** (for example, in kg/s, W, and so on).

Write...
opens the **Select File** dialog box (The Select File Dialog Box (p. 19) and The Select File Dialog Box (Windows) (p. 19)), which you can use to save the reported values to a file.

37.21.2. Force Reports Dialog Box

The **Force Reports** dialog box allows you to compute the forces along a specified vector, moments about a specified center, and the coordinates of the center of pressure for selected wall zones. See Forces on Boundaries (p. 1916) for details.



Controls

Options
contains option buttons that control computation of the forces, moments, or center of pressure.

Forces
enables the computation of the pressure and viscous forces.

Moments
enables the computation of the pressure and viscous moments.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Center of Presure
enables the computation of the average location of the pressure.

Direction Vector
contains the components of the force vector. This label is visible when the **Forces** option button is active.

X,Y,Z
are the components of the force vector along which the forces will be computed.

Moment Center
contains the Cartesian coordinates of the moment center. This label is visible when the **Moments** option button is active.

X,Y,Z
are the Cartesian coordinates of the moment center about which moments will be computed.

Moment Axis
contains the Cartesian coordinates of the moment axis. This label is visible when the **Moments** option button is active.

X,Y,Z
are the Cartesian coordinates of the moment axis about which moments will be computed.

Coordinate
contains the value of the Cartesian coordinate that is fixed. This label is visible when the **Center of Pressure** option button is active.

X,Y,Z
are the Cartesian coordinates, one of which will be fixed.

Value (n)
is the point where the selected Cartesian coordinate will be fixed.

Wall Zones
contains a selectable list of wall zones. The force or moment information is printed for each zone, and then a total force or moment for all the zones is presented.

Save Output Parameter...
opens the Save Output Parameter Dialog Box (p. 2631).

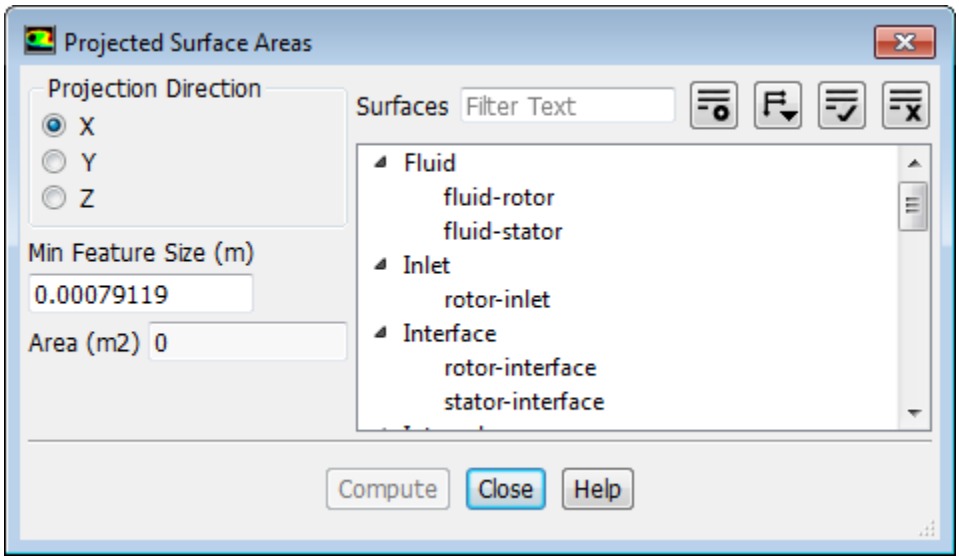
Print
displays (in the console) the pressure, viscous (if appropriate), and total forces or moments, and the pressure, viscous, and total force coefficients along the specified force vector or moment center for the selected wall zones. The center of pressure coordinates will print to the console when the **Center of Pressure** option is activated.

Write...
opens The Select File Dialog Box (p. 19), which you can use to save the reported values to a file.

37.21.3. Projected Surface Areas Dialog Box

The **Projected Surface Areas** dialog box allows you to compute an estimated area of the projection of selected surfaces along the x, y, or z axis. See Projected Surface Area Calculations (p. 1919) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Projection Direction
indicates the direction along which to project the surface.

Min Feature Size
specifies the length of the smallest feature in the geometry that you want to resolve in the area calculation.

Area
displays the computed projected area.

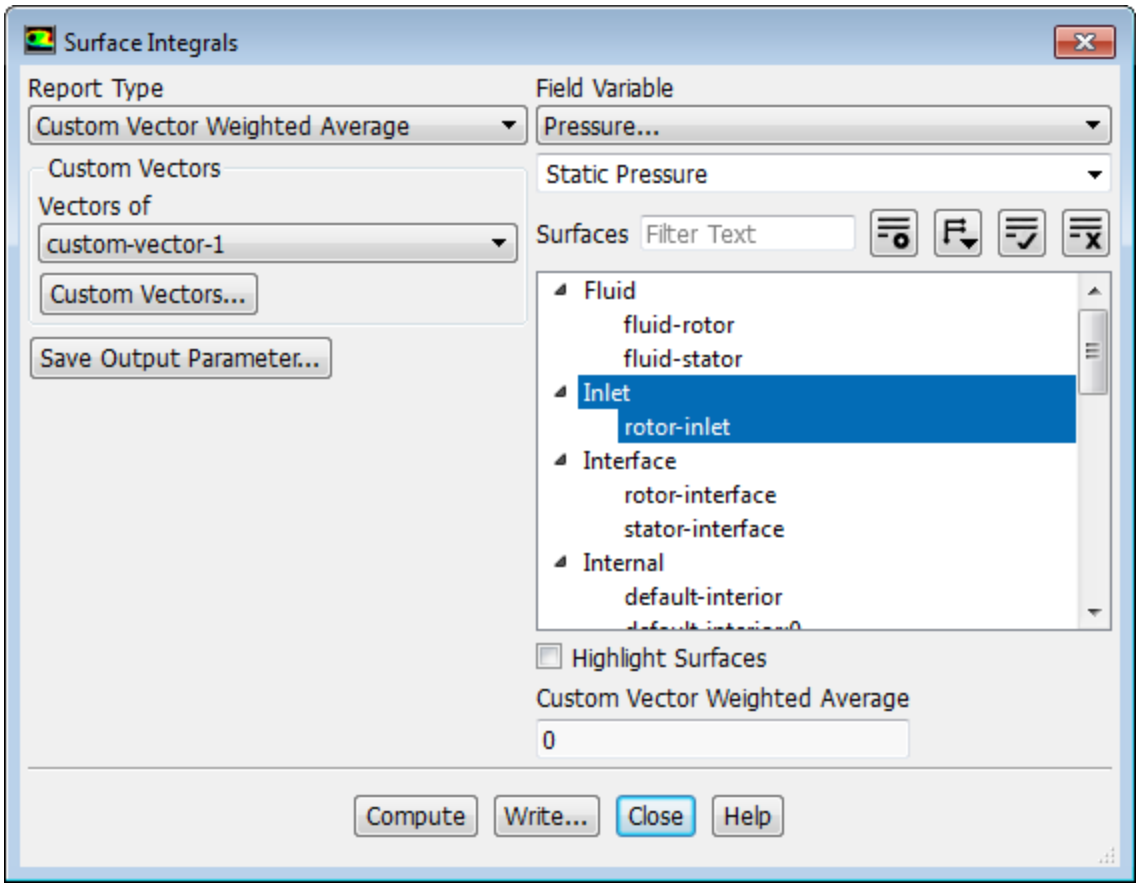
Surfaces
contains a list of existing surfaces. You can select the surface(s) for which the projected area is to be calculated in this list.

Compute
computes the area of the selected surfaces projected along the selected direction. The area will be printed in the **Area** box and in the console window.

37.21.4. Surface Integrals Dialog Box

The **Surface Integrals** dialog box allows you to compute the area, mass and volume flow rate, standard deviation, integral, flow rate, area-weighted average, custom vector-based flux, custom vector flux, custom vector-weighted average, mass-weighted average, sum, facet average, facet minimum/maximum, uniformity index (weighted by area or mass), vertex average, or vertex minimum/maximum quantity of a specified field variable on a selected list of surfaces. See Surface Integration (p. 1920) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Report Type
contains drop-down list that has options that control the method of surface integration.

Area
turns on the computation of the surface area.

Area-Weighted Average
turns on the computation of the area-weighted average on the surface(s).

Custom Vector Based Flux
turns on the computation of the vector-based flow rate of a quantity through the surface(s).

Custom Vector Flux
turns on the computation of the flow rate on a vector through the surface(s).

Custom Vector Weighted Average
turns on the computation of vector-weighted average of a quantity on the surface(s).

Facet Average
turns on the computation of the facet-averaged quantity on the surface(s).

Facet Minimum
turns on the computation of the facet minimum of a quantity on the surface(s).

Facet Maximum
turns on the computation of the facet maximum of a quantity on the surface(s).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Flow Rate
turns on the computation of the flow rate through the surface(s).

Integral
turns on the computation of the integral on the surface(s).

Mass Flow Rate
turns on the computation of the mass flow rate through the surface(s).

Mass-Weighted Average
turns on the computation of the mass-averaged quantity on the surface(s).

Standard Deviation
turns on the computation of the standard deviation of a specified field variable on a surface.

Sum
turns on the computation of the summed quantity on the surface(s).

Uniformity Index - Mass Weighted
turns on the computation of the mass-weighted uniformity index of a quantity on the surface(s).

Uniformity Index - Area Weighted
turns on the computation of the area-weighted uniformity index of a quantity on the surface(s).

Vertex Average
turns on the computation of the vertex-averaged quantity on the surface(s).

Vertex Minimum
turns on the computation of the vertex minimum of a quantity on the surface(s).

Vertex Maximum
turns on the computation of the vertex maximum of a quantity on the surface(s).

Volume Flow Rate
turns on the computation of the volume flow rate through the surface(s).

Custom Vectors
allows you to define custom vectors if you selected **Custom Vector Based Flux**, **Custom Vector Flux**, or **Custom Vector Weighted Average** from the **Report Type** drop-down list.

Field Variable
contains a list of the field variables that can be used in the surface integrations. This option is not active if the **Area**, **Mass Flow Rate**, or **Volume Flow Rate** option is active.

Surfaces
is a selectable list of surfaces.

Area
displays the result of the area summation over all the selected surfaces. This label is visible when **Area** is active.

Area-Weighted Average
displays the result of the area-weighted average computation over all the selected surfaces. This label is visible when **Area-Weighted Average** is active.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Average of Facet Values
displays the result of the facet-averaged computation over all the selected surfaces. This label is visible when **Facet Average** is active.

Minimum of Facet Values
displays the minimum facet value on all the selected surfaces. This label is visible when **Facet Minimum** is active.

Maximum of Facet Values
displays the maximum facet value on all the selected surfaces. This label is visible when **Facet Maximum** is active.

Flow Rate
displays the result of the flow rate computation over all the selected surfaces. This label is visible when **Flow Rate** is active.

Integral
displays the result of the integral computation over all the selected surfaces. This label is visible when **Integral** is active.

Mass Flow Rate
displays the result of the mass flow rate computation over all the selected surfaces. This label is visible when **Mass Flow Rate** is active.

Mass-Weighted Average
displays the result of the mass-averaged computation over all the selected surfaces. This label is visible when **Mass-Weighted Average** is active.

Standard Deviation
displays the result of the standard deviation computation over all the selected surfaces. This label is visible when **Standard Deviation** is active.

Sum of Facet Values
displays the result of the summation over all the selected surfaces. This label is visible when **Sum** is active.

Uniformity Index Mass-Wt.
displays the mass-weighted uniformity index value on all the selected surfaces. This label is visible when **Uniformity Index - Mass Weighted** is active.

Uniformity Index Area-Wt.
displays the area-weighted uniformity index value on all the selected surfaces. This label is visible when **Uniformity Index - Area Weighted** is active.

Average of Surface Vertex Values
displays the result of the vertex-averaged computation over all the selected surfaces. This label is visible when **Vertex Average** is active.

Minimum of Vertex Values
displays the minimum facet value on all the selected surfaces. This label is visible when **Vertex Minimum** is active.

Maximum of Vertex Values
displays the maximum facet value on all the selected surfaces. This label is visible when **Vertex Maximum** is active.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Volumetric Flow Rate
displays the result of the volumetric flow rate computation over all the selected surfaces. This label is visible when **Volume Flow Rate** is active.

Highlight Surfaces
when enabled highlights the surfaces (selected in the **Surface Integrals** dialog box) in the graphics window.

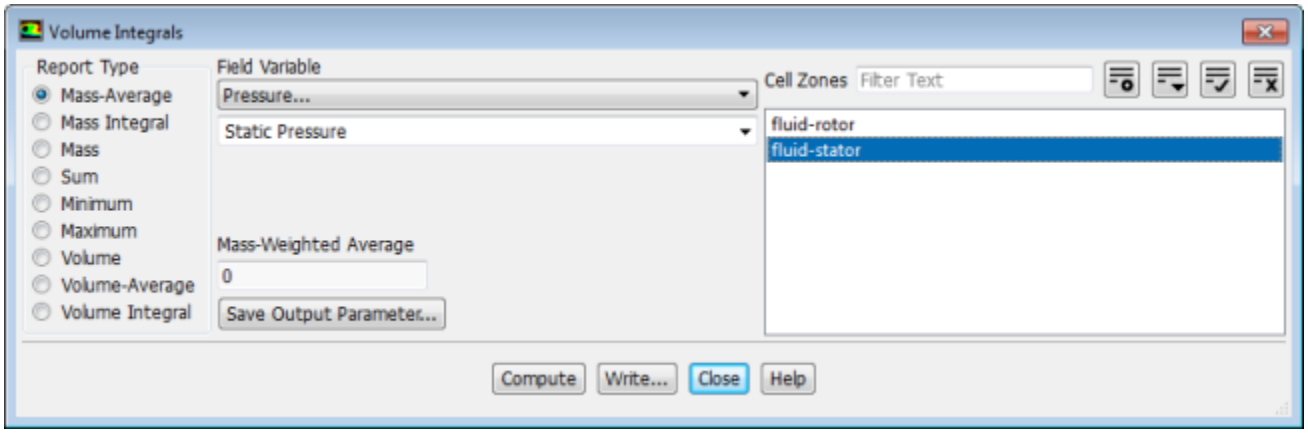
Save Output Parameter...
opens the Save Output Parameter Dialog Box (p. 2631).

Compute
computes the specified integration on the selected surfaces.

Write...
opens The Select File Dialog Box (p. 19), which you can use to save the reported values to a file.

37.21.5. Volume Integrals Dialog Box

The **Volume Integrals** dialog box allows you to compute the volume, sum, maximum, minimum, volume integral, volume-averaged quantity, mass integral, or mass-averaged quantity of a specified field variable on a selected list of cell zones. See Volume Integration (p. 1922) for details.



Controls

Report Type
contains option buttons that control the method of volume integration.

Mass-Average
turns on the computation of the mass-averaged quantity on the cell zone.

Mass Integral
turns on the computation of the mass integral on the cell zone.

Mass
turns on the computation of the total mass in the cell zone for the selected phase.

Sum
turns on the computation of the summation over all cells in the selected zone.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sum*2Pi
computes the summation over all cells in the selected zone of a 2D axisymmetric model and multiplies the result by 2π to give the summation over the revolved domain. This option is only available in 2D axisymmetric cases.

Minimum
computes the minimum value of the selected variable at each cell in the selected zone.

Maximum
computes the maximum value of the selected variable at each cell in the selected zone.

Volume
turns on the computation of the cell zone volume.

Volume Integral
turns on the computation of the volume integral on the cell zone.

Volume-Average
turns on the computation of the volume-weighted average on the cell zone.

Field Variable
contains a list of the field variables that can be used in the sum, volume integral, and average computations. This option is not active if the **Volume** option is active.

Cell Zones
is a selectable list of cell zones.

Total Volume
displays the result of the volume computation over all the selected zones. This label is visible when **Volume** is active.

Sum
displays the result of the summation over all the selected zones. This label is visible when **Sum** is active.

Sum *2Pi
displays the result of the summation over all the selected zones multiplied by 2π. This label is visible when **Sum*2Pi** is active.

Max
displays the result of the maximum value of the selected zone(s). This label is visible when **Maximum** is active.

Min
displays the result of the minimum value of the selected zone(s). This label is visible when **Minimum** is active.

Total Volume Integral
displays the result of the volume-integral computation over all the selected zones. This label is visible when **Volume Integral** is active.

Volume-Weighted Average
displays the result of the volume-averaged computation over all the selected zones. This label is visible when **Volume-Average** is active.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Total Mass-Weighted Integral
displays the result of the mass-integral computation over all the selected zones. This label is visible when **Mass Integral** is active.

Mass-Weighted Average
displays the result of the mass-averaged computation over all the selected zones. This label is visible when **Mass-Average** is active.

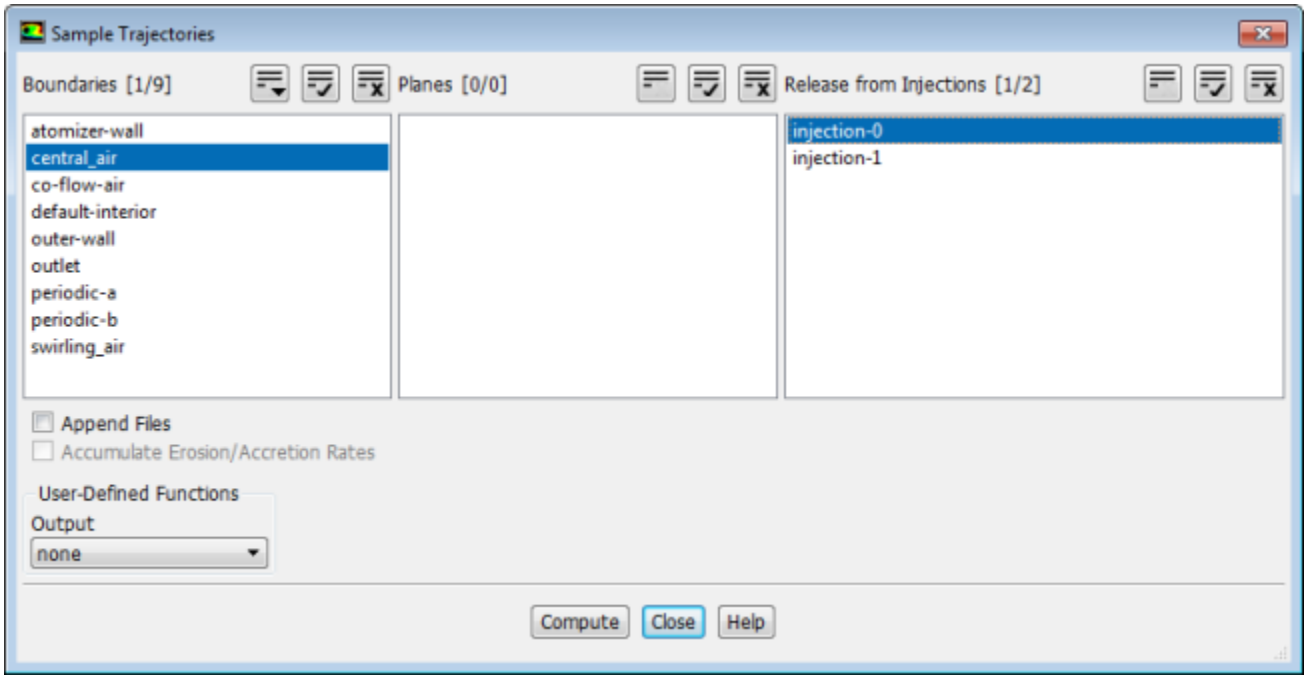
Save Output Parameter...
opens the Save Output Parameter Dialog Box (p. 2631).

Compute
computes the specified integration on the selected zones.

Write...
opens The Select File Dialog Box (p. 19), which you can use to save the reported values to a file.

37.21.6. Sample Trajectories Dialog Box

The **Sample Trajectories** dialog box allows the writing of particle states (position, velocity, diameter, temperature, and mass flow rate) at various boundaries and planes (lines in 2D). See Sampling of Trajectories (p. 1355) for details about the items below.



Controls

Boundaries
lists boundaries that can be chosen as the surfaces at which samples will be written.

Lines
(in 2D) lists lines that can be chosen as the surfaces at which samples will be written.

Planes
(in 3D) lists planes that can be chosen as the surfaces at which samples will be written.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release From Injections
lists injections from which the injection to be tracked is chosen.

Append Files
causes the results of multiple calculations to be appended to a single file.

Accumulate Erosion/Accretion Rates
causes erosion and accretion rates to be accumulated for repeated trajectory calculations.

User-Defined Functions
allow control of the format and the information written for the sample output.

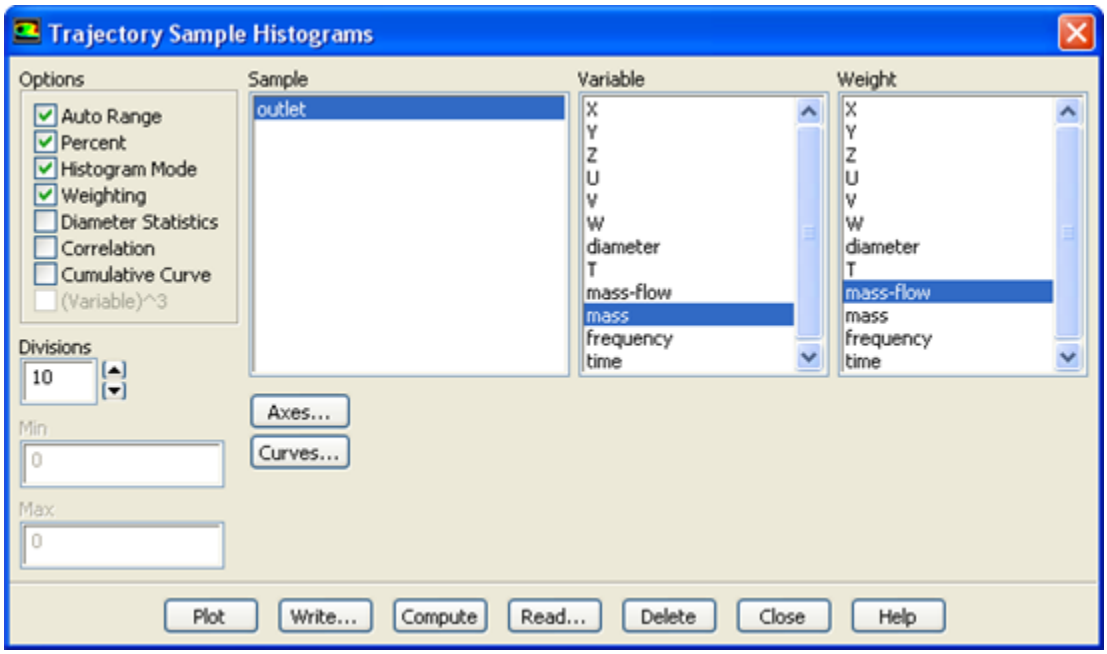
Output
contains a drop-down list of available user-defined functions.

Compute
causes the particles to be tracked and their status to be written to files when they encounter selected surfaces. This button will not appear for unsteady particle tracking.

Start
initiates sampling for unsteady particle tracking. This button will replace the **Compute** button if you are performing unsteady particle tracking. After you click it, it will change to the **Stop** button. Click **Stop** to stop the sampling. (The label will change back to **Start**.)

37.21.7. Trajectory Sample Histograms Dialog Box

The **Trajectory Sample Histograms** dialog box allows the plotting of histograms from sample files created in the Sample Trajectories Dialog Box (p. 2622). See Histogram Reporting of Samples (p. 1356) for details about the items below.



Controls

Options
contains check buttons for histogram options.

Auto Range
toggles between automatic and manual settings of the histogram range.

Percent
causes the plot to indicate the percent of particles. Deselecting this will result in the actual number of particles being plotted.

Histogram Mode
allows you to display the histogram with or without bars.

Weighting
allows you to apply a weight to the data sample.

Diameter Statistics
allows you to display a summary of diameter statistics for a selected variable in the console window.

Correlation
allows you to choose correlations between how one particle variable depends on another particle variable.

Cumulative Curve
allows you to compute a cumulative distribution curve for a selected variable.

(Variable^3)
allows you to plot the cumulative mass distribution for a constant particle density using the particle diameter.

Divisions
sets the number of "bins" in the histogram.

Sample
lists the data samples that have been read in.

Variable
lists the fields variables available in the selected sample.

Weight
lists the weighted fields variables available in the selected sample.

Correlation
lists the sampled variables, allowing you to choose the correlation variable.

Min
displays the minimum value of the variable selected in the **Variable** list. If **Auto Range** is off, you can set the minimum by typing a new value.

Max
displays the maximum value of the variable selected in the **Fields** list. If **Auto Range** is off, you can set the maximum by typing a new value.

Axes...
opens the Axes Dialog Box (p. 2608), which allows you to customize the histogram axes.

Curves...
opens the Curves Dialog Box (p. 2610), which allows you to customize the curves used in the histogram.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plot
displays the histogram in the active graphics window.

Write...
stores the histograms in an XY-plot file format

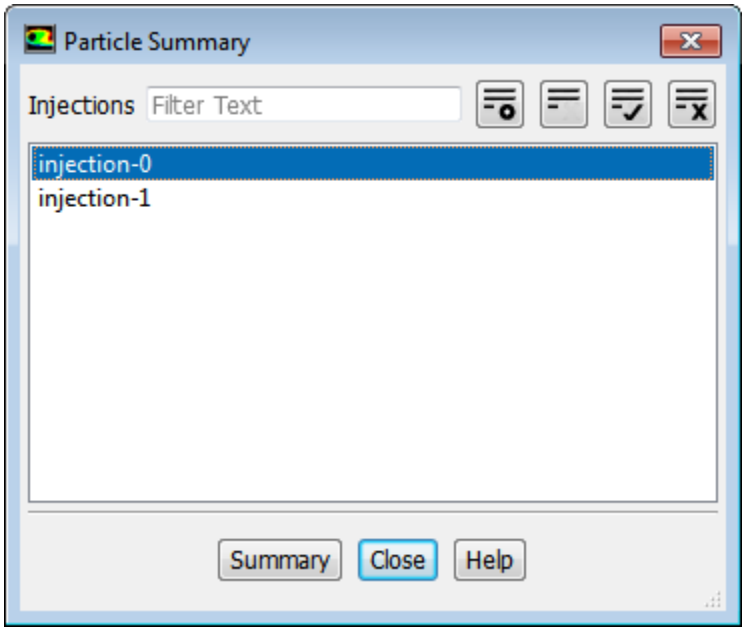
Compute
updates the **Min** and **Max** values.

Read...
opens The Select File Dialog Box (p. 19), where you can select a sample file to be read.

Delete
removes the sample selected in the **Sample** list.

37.21.8. Particle Summary Dialog Box

The **Particle Summary** dialog box allows you to report a summary for particle injections. See Summary Reporting of Current Particles (p. 1358) for details about the items below.



Controls

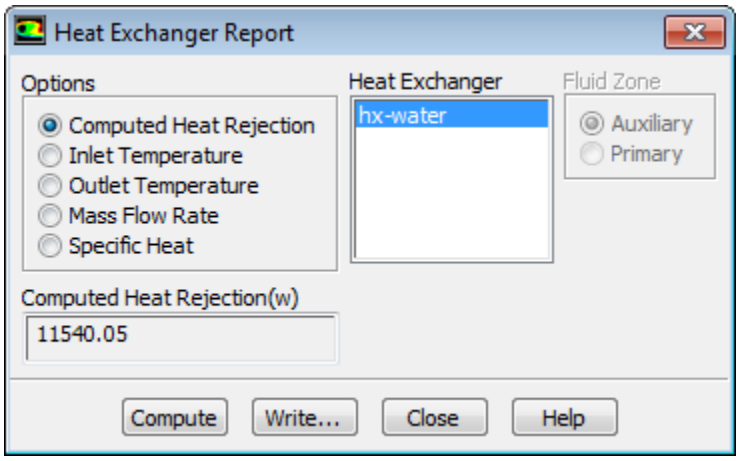
Injections
lists the particle injection(s) for which you can generate a summary.

Summary
prints the injection summary in the console window.

37.21.9. Heat Exchanger Report Dialog Box

The **Heat Exchanger Report** dialog box allows you to report a summary for heat exchangers. See Postprocessing for the Heat Exchanger Model (p. 950) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options
lists the available reporting options for the heat exchanger.

Computed Heat Rejection
allows you to report the heat rejection calculated from the heat exchanger.

Inlet Temperature
allows you to report the inlet temperature for both the primary and the auxiliary heat exchanger fluid.

Outlet Temperature
allows you to report the outlet temperature for both the primary and the auxiliary heat exchanger fluid.

Mass Flow Rate
allows you to report the mass flow rate for both the primary and the auxiliary heat exchanger fluid.

Specific Heat
allows you to report the specific heat for both the primary and the auxiliary heat exchanger fluid.

Result
displays the results of the calculations once the **Compute** button is selected.

Heat Exchanger
displays a list of heat exchanger fluid zones

Fluid Zone
(not available when **Computed Heat Rejection** option is selected.) allows you to report quantities for either the primary or auxiliary fluid zones.

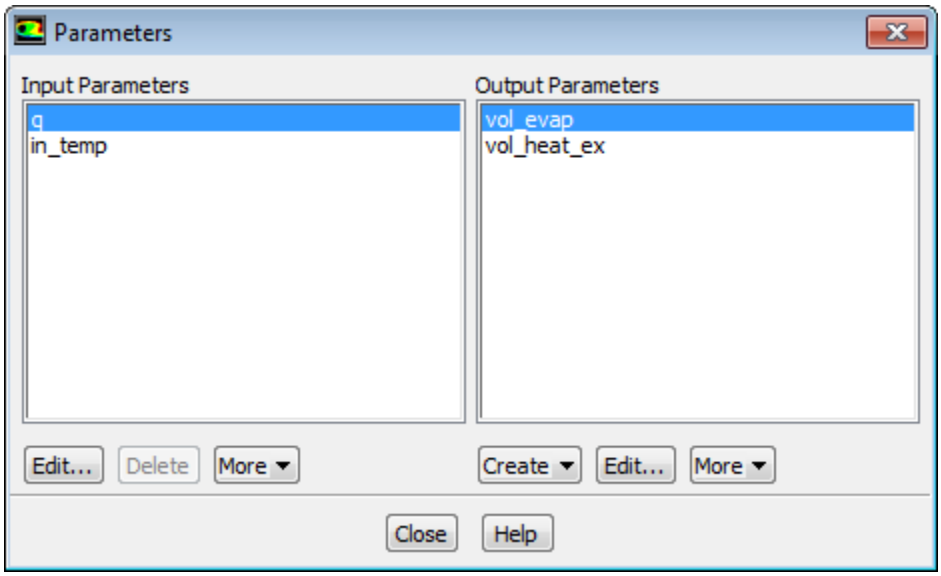
Compute
computes the specified quantity on the selected zones.

Write...
opens The Select File Dialog Box (p. 19), which you can use to save the reported values to a file.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.21.10. Parameters Dialog Box

The **Parameters** dialog box allows you to create input parameters (in Scheme or user-defined function procedures only) and output parameters, which allow you to compare reporting values for different cases, or include reporting values in the function minimized by the mesh morpher/optimizer. See Creating Output Parameters (p. 1909) for details about the items below.



Controls

Input Parameters
contains a list of existing input parameters. See Defining and Viewing Parameters (p. 243) for details about creating input parameters.

Edit...
opens the Input Parameter Properties Dialog Box (p. 2631).

> **Important**
>
> When using ANSYS Fluent in ANSYS Workbench, parameters are not editable, so the **Edit...** button becomes the **View...** button. This opens the Input Parameter Properties Dialog Box (p. 2631) where the parameter properties can only be viewed. For more information, see the separate ANSYS Fluent in **Workbench** User's Guide.

Delete
removes the selected input parameter from the list of **Input Parameters**.

More

contains a drop-down list that allows more advanced use of input parameters. Available options include:

Use in Scheme Procedure
opens the Use Input Parameter in Scheme Procedure Dialog Box (p. 2628).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use in UDF
opens the Use Input Parameter for UDF Dialog Box (p. 2629).

Output Parameters
contains a list of existing output parameters.

Create
opens the Report Definitions Dialog Box (p. 2785), allowing you to create additional output parameters based on report definitions.

Edit...
opens the dialog box corresponding to the selected item in the **Output Parameters** list.

More
contains a drop-down list containing additional tasks that you can perform, including:

Delete
removes the selected output parameter from the list of **Output Parameters**.

Print to Console
reports values to the console window. If you select multiple output parameters, then the output includes values from multiple output parameters.

Print All to Console
outputs the values from all output parameters to the console window.

Write...
allows you to store the output to a file. A dialog box is displayed allowing you to provide a file name.

Write All...
prompts you for a file name and then writes the values for all of the output parameters to a file.

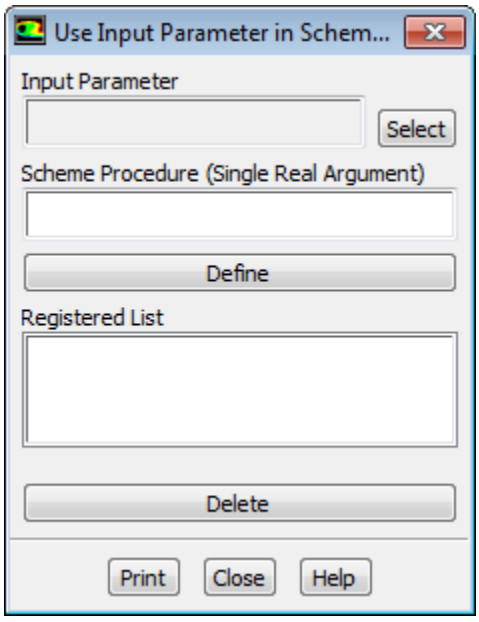
Note

ANSYS Fluent automatically creates generic default names for new input and output parameters (for example, `parameter-1`, `parameter-2`, `parameter-3`, and so on) If a parameter is deleted, the default name is not reused. For example, if you have `parameter-1`, `parameter-2`, and `parameter-3`, then delete `parameter-2` and create a new parameter, the default name for the new parameter will be `parameter-4`.

37.21.11. Use Input Parameter in Scheme Procedure Dialog Box

The **Use Input Parameter in Scheme Procedure** dialog box allows you to use the input parameter in a Scheme procedure. In turn, this Scheme procedure can also use text user interface (TUI) commands. For more information, see Working With Advanced Parameter Options (p. 247).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Input Parameter
contains the name of an input parameter.

Select
opens the Select Input Parameter Dialog Box (p. 2384) where an input parameter can be chosen.

Scheme Procedure (Single Real Argument)
contains a procedure name (if already defined using a Scheme file) or procedure body written in the Scheme language using `lambda`. This procedure should receive one real argument.

Define
registers the selected input parameter.

Registered List
contains a list of registered input parameters.

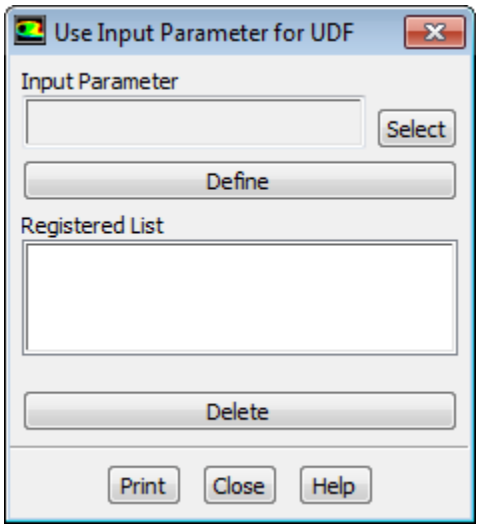
Delete
removes a selected registered input parameter from the list.

Print
prints details of the registered input parameters.

37.21.12. Use Input Parameter for UDF Dialog Box

The **Use Input Parameter for UDF** dialog box allows you to select the input parameters that can be used inside a user-defined function (UDF) during calculations.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Input Parameter
contains the name of an input parameter.

Select
opens the Select Input Parameter Dialog Box (p. 2384) where an input parameter can be chosen.

Define
registers the selected input parameter.

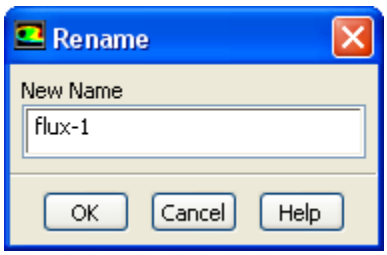
Registered List
contains a list of registered input parameters.

Delete
removes a selected registered input parameter from the list.

Print
prints the ID of the registered input parameters. The ID values are used to access the value of the parameter in the user-defined function.

37.21.13. Rename Dialog Box

The **Rename** dialog box allows you to change the name of the output parameter that you created.



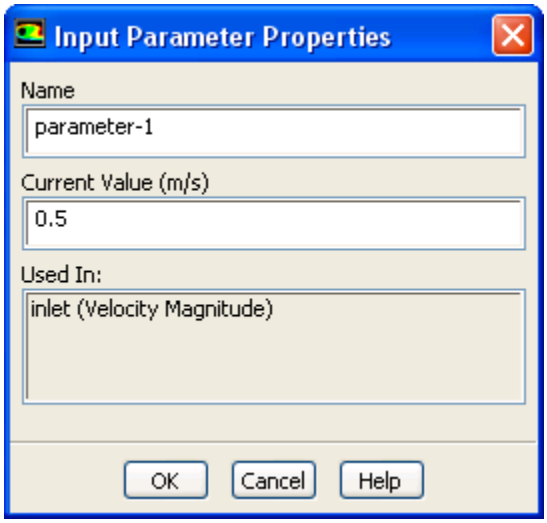
Controls

New Name
contains the new name of the output parameter.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.21.14. Input Parameter Properties Dialog Box

The **Input Parameter Properties** dialog box allows you to create input parameters, which allow you to compare reporting values for different cases. See Creating Output Parameters (p. 1909) for details about the items below.



Controls

Name
contains the name of the input parameter.

Current Value
contains the current value of the input parameter.

Used In
indicates the cell zone or boundary condition where the input parameter is currently used.

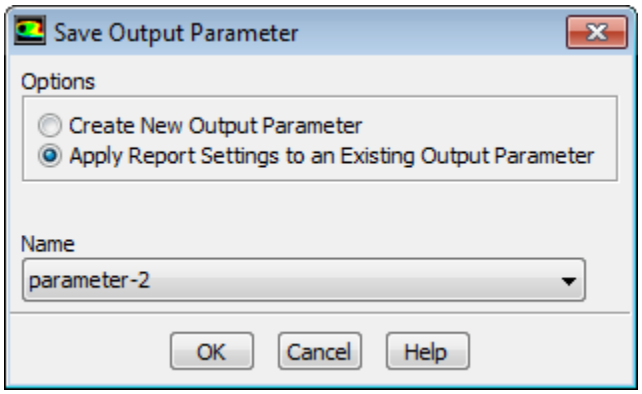
Note

ANSYS Fluent automatically creates generic default names for new input and output parameters (for example, `parameter-1`, `parameter-2`, `parameter-3`, and so on) If a parameter is deleted, the default name is not reused. For example, if you have `parameter-1`, `parameter-2`, and `parameter-3`, then delete `parameter-2` and create a new parameter, the default name for the new parameter will be `parameter-4`.

37.21.15. Save Output Parameter Dialog Box

The **Save Output Parameter** dialog box allows you to save specific output parameters that allow you to compare reporting values for different cases. See Creating Output Parameters (p. 1909) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options
contains options for saving your output parameters.

Create New Output Parameter
allows you to create a new output parameter.

Apply Report Settings to an Existing Output Parameter
allows you to overwrite an existing output parameter of the same type.

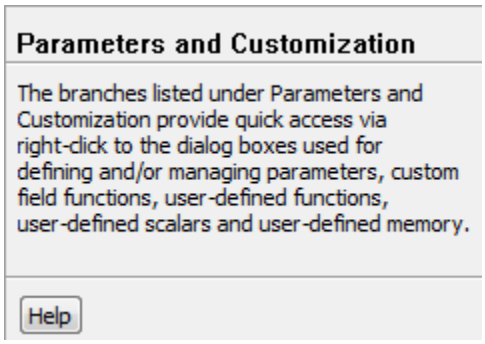
Name
contains the name for the current output parameter.

Note

ANSYS Fluent automatically creates generic default names for new input and output parameters (for example, `parameter-1`, `parameter-2`, `parameter-3`, and so on) If a parameter is deleted, the default name is not reused. For example, if you have `parameter-1`, `parameter-2`, and `parameter-3`, then delete `parameter-2` and create a new parameter, the default name for the new parameter will be `parameter-4`.

37.22. Parameters and Customization Task Page

The **Parameters and Customization** task page introduces you to the parameterization and customization tools available for use in solving your CFD simulation using ANSYS Fluent.



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 38: Ribbon Reference Guide

This reference guide provides information about the ribbon in ANSYS Fluent.

38.1. File Ribbon Tab
38.2. Dialog Boxes Available from the Ribbon

38.1. File Ribbon Tab

For additional information, see the following sections:

38.1.1. File/Read/Mesh...
38.1.2. File/Read/Case...
38.1.3. File/Read/Data...
38.1.4. File/Read/Case & Data...
38.1.5. File/Read/PDF...
38.1.6. File/Read/ISAT Table...
38.1.7. File/Read/DTRM Rays...
38.1.8. File/Read/View Factors...
38.1.9. File/Read/Profile...
38.1.10. File/Read/Scheme...
38.1.11. File/Read/Journal...
38.1.12. File/Write/Case...
38.1.13. File/Write/Data...
38.1.14. File/Write/Case & Data...
38.1.15. File/Write/PDF...
38.1.16. File/Write/ISAT Table...
38.1.17. File/Write/Flamelet...
38.1.18. File/Write/Surface Clusters...
38.1.19. File/Write/Profile...
38.1.20. File/Write/Autosave...
38.1.21. File/Write/Boundary Mesh...
38.1.22. File/Write/Start Journal...
38.1.23. File/Write/Stop Journal
38.1.24. File/Write/Start Transcript...
38.1.25. File/Write/Stop Transcript
38.1.26. File/Import/ABAQUS/Input File...
38.1.27. File/Import/ABAQUS/Filbin File...
38.1.28. File/Import/ABAQUS/ODB File...
38.1.29. File/Import/CFX/Definition File...
38.1.30. File/Import/CFX/Result File...
38.1.31. File/Import/CGNS/Mesh...
38.1.32. File/Import/CGNS/Data...
38.1.33. File/Import/CGNS/Mesh & Data...
38.1.34. File/Import/EnSight...
38.1.35. File/Import/FIDAP...
38.1.36. File/Import/GAMBIT...
38.1.37. File/Import/HYPERMESH ASCII...
38.1.38. File/Import/I-deas Universal...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.1.39. File/Import/LSTC/Input File...
38.1.40. File/Import/LSTC/State File...
38.1.41. File/Import/Marc POST...
38.1.42. File/Import/Mechanical APDL/Input File...
38.1.43. File/Import/Mechanical APDL/Result File...
38.1.44. File/Import/NASTRAN/Bulkdata File...
38.1.45. File/Import/NASTRAN/Op2 File...
38.1.46. File/Import/PATRAN/Neutral File...
38.1.47. File/Import/PLOT3D/Grid File...
38.1.48. File/Import/PLOT3D/Result File...
38.1.49. File/Import/PTC Mechanica Design...
38.1.50. File/Import/Tecplot...
38.1.51. File/Import/Fluent 4 Case File...
38.1.52. File/Import/PreBFC File...
38.1.53. File/Import/Partition/Metis...
38.1.54. File/Import/Partition/Metis Zone...
38.1.55. File/Import/CHEMKIN Mechanism...
38.1.56. File/Export/Solution Data...
38.1.57. File/Export/Particle History Data...
38.1.58. File/Export/During Calculation/Solution Data...
38.1.59. File/Export/During Calculation/Particle History Data...
38.1.60. File/Export to CFD-Post...
38.1.61. File/Solution Files...
38.1.62. File/Interpolate...
38.1.63. File/FSI Mapping/Volume...
38.1.64. File/FSI Mapping/Surface...
38.1.65. File/Save Picture...
38.1.66. File/Data File Quantities...
38.1.67. File/Batch Options...
38.1.68. File/Exit

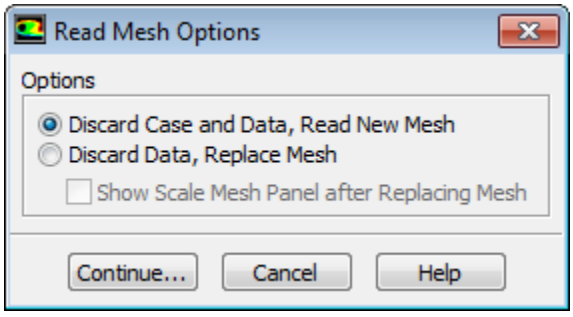
38.1.1. File/Read/Mesh...

The **File/Read/Mesh...** ribbon tab item opens the Read Mesh Options Dialog Box (p. 2634), which allows you to read or replace a mesh.

38.1.1.1. Read Mesh Options Dialog Box

The **Read Mesh Options** dialog box is used to read or replace a mesh. See Reading Mesh Files (p. 44) for more information.

File → Read → Mesh...



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Options

allows you to read in a new mesh or replace the existing mesh.

Discard Case And Data, Read New Mesh

results in both the case and data files being discarded when reading in a new mesh.

Discard Data, Replace Mesh

results in the data file being discarded when replacing an existing mesh.

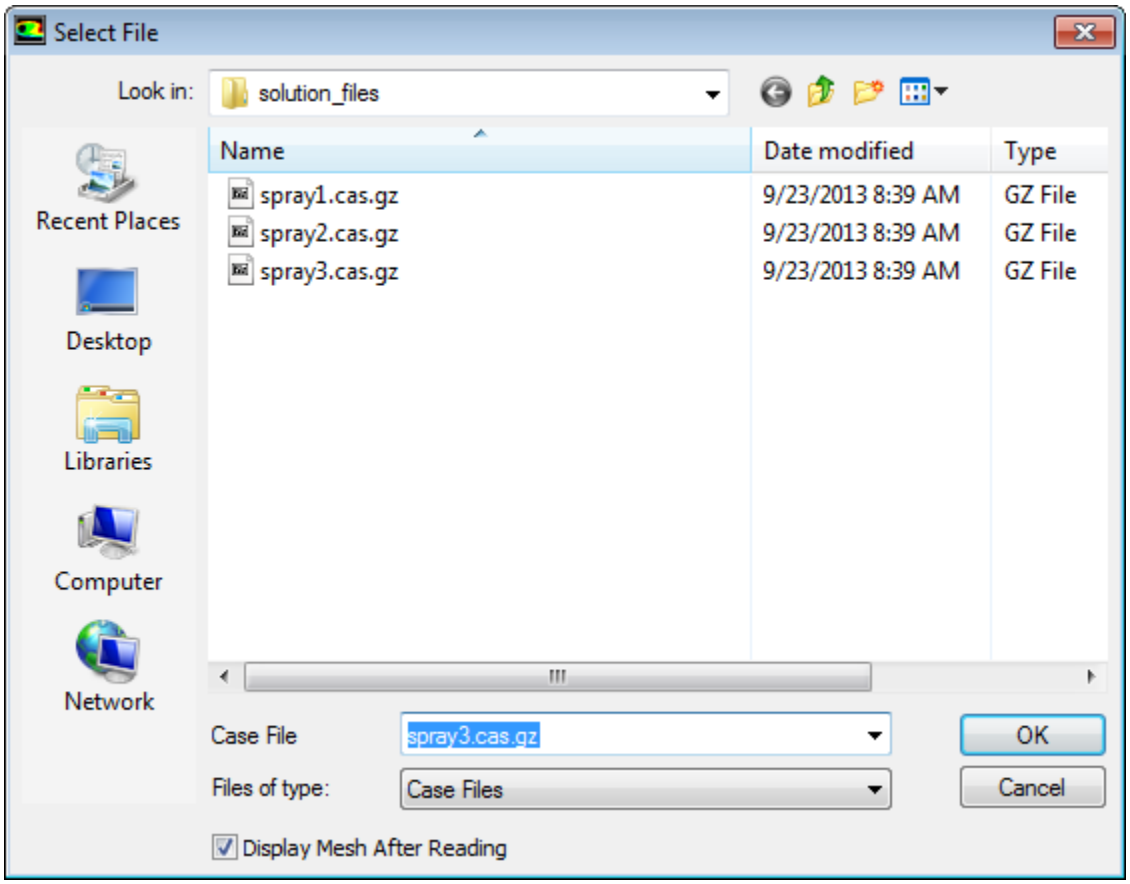
Show Scale Mesh Panel After Replacing Mesh

gives you the option to have the Scale Mesh Dialog Box (p. 2162) appear automatically for you to check or scale your mesh.

38.1.2. File/Read/Case...

The **File/Read/Case...** ribbon tab item is used to read in an ANSYS Fluent case file (extension `.cas`), or a mesh file (extension `.msh`, `.grd`, `.MSH`, or `.GRD`) that has been saved in the native format for ANSYS Fluent. See Reading Mesh Files (p. 44) and Reading and Writing Case and Data Files (p. 45) for details.

The **File/Read/Case...** ribbon tab item opens The Select File Dialog Box (p. 19), which allows you to select the appropriate file to be read.



38.1.3. File/Read/Data...

The **File/Read/Data...** ribbon tab item is used to read in an ANSYS Fluent data file (which has a `.dat` extension) or parallel data file (which has a `.pdat` extension). This item will not be available until you read in a case or mesh file. See Reading and Writing Data Files (p. 47) and Reading and Writing Parallel Data Files (p. 51) for details.

The **File/Read/Data...** ribbon tab item opens The Select File Dialog Box (p. 19), which allows you to select the appropriate file to be read.

38.1.4. File/Read/Case & Data...

The **File/Read/Case & Data...** ribbon tab item is used to read in an ANSYS Fluent case file and the corresponding data file (for example, `myfile.cas` and `myfile.dat`) together. See Reading and Writing Case and Data Files Together (p. 47) for details.

The **File/Read/Case & Data...** ribbon tab item opens The Select File Dialog Box (p. 19), which allows you to select the appropriate files to be read. Select the appropriate case file, and the corresponding data file (that is, the file having the same name with a `.dat` extension) will also be read in.

38.1.5. File/Read/PDF...

The **File/Read/PDF...** ribbon tab item is used to read a PDF file (extension `.pdf`) created by ANSYS Fluent for use with the non-premixed or partially premixed combustion model. This item is available only when the non-premixed or partially premixed combustion model has been enabled. See Saving the Look-Up Tables (p. 1068) for details.

38.1.6. File/Read/ISAT Table...

The **File/Read/ISAT Table...** ribbon tab item is used to read an ISAT table (extension `.isat`) for use with the Composition PDF Transport model. This item is available only when the Composition PDF Transport model has been enabled. See Using ISAT Efficiently (p. 1130) for details.

38.1.7. File/Read/DTRM Rays...

The **File/Read/DTRM Rays...** ribbon tab item is used to read a ray file (extension `.ray`) created by ANSYS Fluent for use with the DTRM (radiation model). This item is available only when the DTRM has been enabled. See Writing and Reading the DTRM Ray File (p. 850) for details.

38.1.8. File/Read/View Factors...

The **File/Read/View Factors...** ribbon tab item is used to read in a view factor file for use with the surface-to-surface (S2S) radiation model. This item is available only when the S2S model has been enabled. See Reading View Factors into ANSYS Fluent (p. 863) for details.

38.1.9. File/Read/Profile...

The **File/Read/Profile...** ribbon tab item opens the **Select File** dialog box for reading profiles (Reading Profile Files (p. 54)), It is used to read a cell zone or boundary condition profile file (extension `.prof`). A profile file defines profiles that can be used to specify flow conditions for a cell zone or a boundary. See Profiles (p. 435) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.1.10. File/Read/Scheme...

The **File/Read/Scheme...** ribbon tab item is used to read in a Scheme source file (extension `.scm`). See Reading Scheme Source Files (p. 57) for details.

38.1.11. File/Read/Journal...

The **File/Read/Journal...** ribbon tab item is used to read in a journal file (extension `.jou`) containing a sequence of ANSYS Fluent commands. You can create a journal file using the **File/Write/Journal...** ribbon tab item. See Creating and Reading Journal Files (p. 57) for details.

38.1.12. File/Write/Case...

The **File/Write/Case...** ribbon tab item is used to save an ANSYS Fluent case file. See Reading and Writing Case Files (p. 46) for details.

The **File/Write/Case...** ribbon tab item opens The Select File Dialog Box (p. 19), which allows you to save the file with a name of choice. The dialog box is similar to the **Select File** dialog box for reading files, except that it has an additional option for writing binary files.

Important

- When ANSYS Fluent writes a case file, the `.cas` extension is added to the file name specified, unless the name already ends with `.cas`.
- You can also write a compressed file by appending `.gz` or `.Z` to the file name. (See Reading and Writing Compressed Files (p. 41) for details about file compression.)

38.1.13. File/Write/Data...

The **File/Write/Data...** ribbon tab item is used to save an ANSYS Fluent data file (which has a `.dat` extension) or parallel data file (which has a `.pdat` extension). See Reading and Writing Data Files (p. 47) and Reading and Writing Parallel Data Files (p. 51) for details.

Important

- When ANSYS Fluent writes a data file, the `.dat` extension is added to the file name specified, unless the name already ends with `.dat` or `.pdat`.
- You can compress data files by appending `.gz` or `.Z` to the file name (see Reading and Writing Compressed Files (p. 41) for details about file compression). To compress parallel data files (that is, files saved with a `.pdat` extension), you must use asynchronous file compression (see Reading and Writing Case and Data Files (p. 45)).

38.1.14. File/Write/Case & Data...

The **File/Write/Case & Data...** ribbon tab item is used to save an ANSYS Fluent case file and data file at the same time (for example, `myfile.cas` and `myfile.dat`). See Reading and Writing Case and Data Files Together (p. 47) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Enter the name of the case file in the text entry box in the **Select File** dialog box and the corresponding data file (same file name, but with a `.dat` extension) will also be written.

38.1.15. File/Write/PDF...

The **File/Write/PDF...** ribbon tab item is used to write a PDF file after computing the look-up tables using the non-premixed or partially premixed combustion model in ANSYS Fluent. This item is available only when the non-premixed or partially premixed combustion model has been enabled. See Saving the Look-Up Tables (p. 1068) for details.

38.1.16. File/Write/ISAT Table...

The **File/Write/ISAT Table...** ribbon tab item is used to write an ISAT table. See Using ISAT Efficiently (p. 1130) for details.

38.1.17. File/Write/Flamelet...

The **File/Write/Flamelet...** ribbon tab item is used to write a flamelet file generated using the non-premixed or partially premixed combustion model in ANSYS Fluent. This item is available only when the non-premixed or partially premixed combustion model has been enabled. See The Diffusion Flamelet Models Theory in the Theory Guide for details.

38.1.18. File/Write/Surface Clusters...

The **File/Write/Surface Clusters...** ribbon tab item is used to set parameters related to surface clusters and view factors for the surface-to-surface radiation model. It opens the View Factors and Clustering Dialog Box (p. 2194). The dialog box that you open using the **File/Write/Surface Clusters...** ribbon tab item is different than the one opened via the **Settings...** button in the **Radiation Model** dialog box, in that it saves the settings you specify to a file that can be used to calculate the view factors outside of ANSYS Fluent (see Computing View Factors Outside ANSYS Fluent (p. 862)). When you click **OK**, The Select File Dialog Box (p. 19) will open so that you can specify a name for the file where ANSYS Fluent should save the settings.

38.1.19. File/Write/Profile...

The **File/Write/Profile...** ribbon tab item opens the Write Profile Dialog Box (p. 2388).

38.1.20. File/Write/Autosave...

The **File/Write/Autosave...** ribbon tab item opens the Autosave Dialog Box (p. 2527).

38.1.21. File/Write/Boundary Mesh...

The **File/Write/Boundary Mesh...** ribbon tab item is used to write the boundary zones (surface mesh) of the domain to a file. You can then read this file into the meshing mode of Fluent or GAMBIT to produce an improved volume mesh. See Writing a Boundary Mesh (p. 56) for details.

38.1.22. File/Write/Start Journal...

The **File/Write/Start Journal...** ribbon tab item is used to start the recording of subsequent ANSYS Fluent commands to a journal file. You can read this journal file back into Fluent later (using the

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File/Read/Journal... ribbon tab item) to automate the execution of the recorded commands. See Creating and Reading Journal Files (p. 57) for details.

38.1.23. File/Write/Stop Journal

The **File/Write/Stop Journal** ribbon tab item replaces the **File/Write/Start Journal...** ribbon tab item after the recording of a journal file has begun. The **File/Write/Stop Journal** ribbon tab item is used to end the journal recording. See Creating and Reading Journal Files (p. 57) for details.

38.1.24. File/Write/Start Transcript...

The **File/Write/Start Transcript...** ribbon tab item is used to start the recording of a transcript file containing all input to and output from Fluent. (You cannot read a transcript file back into Fluent.) See Creating Transcript Files (p. 59) for details.

38.1.25. File/Write/Stop Transcript

The **File/Write/Stop Transcript** ribbon tab item replaces the **File/Write/Start Transcript...** ribbon tab item after the recording of a transcript file has begun. The **File/Write/Stop Transcript** ribbon tab item is used to end the transcript recording. See Creating Transcript Files (p. 59) for details.

38.1.26. File/Import/ABAQUS/Input File...

The **File/Import/ABAQUS/Input File...** ribbon tab item is used to import an ABAQUS input file (extension `.inp`), which contains the input description of a finite element model for the ABAQUS finite element program. See ABAQUS Files (p. 62) for details.

38.1.27. File/Import/ABAQUS/Filbin File...

The **File/Import/ABAQUS/Filbin File...** ribbon tab item is used to import an ABAQUS filbin file (extension `.fil`), which contains the finite element model and results data. See ABAQUS Files (p. 62) for details.

38.1.28. File/Import/ABAQUS/ODB File...

The **File/Import/ABAQUS/ODB File...** ribbon tab item is used to import an ABAQUS ODB file with a `.odb` extension. See ABAQUS Files (p. 62) for details.

38.1.29. File/Import/CFX/Definition File...

The **File/Import/CFX/Definition File...** ribbon tab item is used to import a CFX definition file (extension `.def`), which contains mesh information to be read into Fluent. See CFX Files (p. 62) for details.

38.1.30. File/Import/CFX/Result File...

The **File/Import/CFX/Result File...** ribbon tab item is used to import a CFX result file (extension `.res`). See CFX Files (p. 62) for details.

38.1.31. File/Import/CGNS/Mesh...

The **File/Import/CGNS/Mesh...** ribbon tab item is used to read in a CGNS-format mesh file (extension `.cgns`). See Meshes and Data in CGNS Format (p. 63) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.1.32. File/Import/CGNS/Data...

The **File/Import/CGNS/Data...** ribbon tab item is used to read in a CGNS-format data file. See Meshes and Data in CGNS Format (p. 63) for details.

38.1.33. File/Import/CGNS/Mesh & Data...

The **File/Import/CGNS/Mesh & Data...** ribbon tab item is used to read in a set of CGNS-format mesh and data files. See Meshes and Data in CGNS Format (p. 63) for details.

38.1.34. File/Import/EnSight...

The **File/Import/EnSight...** ribbon tab item is used to import EnSight files (extension `.encas` or `.case`). See EnSight Files (p. 64) for details.

38.1.35. File/Import/FIDAP...

The **File/Import/FIDAP...** ribbon tab item is used to import an ANSYS FIDAP neutral file (extension `.FDNEUT` or `.unv`). See ANSYS FIDAP Neutral Files (p. 64) for details.

38.1.36. File/Import/GAMBIT...

The **File/Import/GAMBIT...** ribbon tab item is used to read in a neutral file from GAMBIT. See GAMBIT and GeoMesh Mesh Files (p. 64) for details.

38.1.37. File/Import/HYPERMESH ASCII...

The **File/Import/HYPERMESH ASCII...** ribbon tab item is used to import a **HYPERMESH ASCII** file (extension `.hm`, `.hma`, or `.hmascii`). See HYPERMESH ASCII Files (p. 65) for details.

38.1.38. File/Import/I-deas Universal...

The **File/Import/I-deas Universal...** ribbon tab item is used to import an I-deas Universal file, which contains mesh information and zone types to be read into Fluent. See I-deas Universal Files (p. 65) for details.

38.1.39. File/Import/LSTC/Input File...

The **File/Import/LSTC/Input File...** ribbon tab item is used to import an LSTC input file (extension `.k`, `.key`, or `.dyn`), which contains the input description of a finite element model for the LS-DYNA finite element program. See LSTC Files (p. 65) for details.

38.1.40. File/Import/LSTC/State File...

The **File/Import/LSTC/State File...** ribbon tab item is used to import an LSTC state file (extension `.d3plot`), which contains control data, geometry data, and state data. See LSTC Files (p. 65) for details.

38.1.41. File/Import/Marc POST...

The **File/Import/Marc POST...** ribbon tab item is used to import a Marc POST file generated using the MSC Marc finite element program. See Marc POST Files (p. 65) for details

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.1.42. File/Import/Mechanical APDL/Input File...

The **File/Import/Mechanical APDL/Input File...** ribbon tab item is used to import a Mechanical APDL input file (extensions `.ans`, `neu`, `.cdb`, or `.prep7`), which contains mesh information to be read into Fluent. See Mechanical APDL Files (p. 66) for details.

38.1.43. File/Import/Mechanical APDL/Result File...

The **File/Import/Mechanical APDL/Result File...** ribbon tab item is used to import a Mechanical APDL result file (extension `.rfl`, `.rst`, `.rth`, or `.rmg`). See Mechanical APDL Files (p. 66) for details.

38.1.44. File/Import/NASTRAN/Bulkdata File...

The **File/Import/NASTRAN/Bulkdata File...** ribbon tab item is used to import a NASTRAN Bulkdata file (extension `.nas`, `.dat`, or `.bdf`), which contains mesh information to be read into Fluent. See NASTRAN Files (p. 66) for details.

38.1.45. File/Import/NASTRAN/Op2 File...

The **File/Import/NASTRAN/Op2 File...** ribbon tab item is used to import a NASTRAN Op2 file (extension `.op2`), which is an output binary data file containing data used in the NASTRAN finite element program. See NASTRAN Files (p. 66) for details.

38.1.46. File/Import/PATRAN/Neutral File...

The **File/Import/PATRAN/Neutral File...** ribbon tab item is used to read in a PATRAN Neutral file (extension `.neu`, `.out`, or `.pat`) zoned by named components (that is, with the nodes placed into zones based on group name). The PATRAN neutral file contains mesh information to be read into Fluent. See PATRAN Neutral Files (p. 67) for details.

38.1.47. File/Import/PLOT3D/Grid File...

The **File/Import/PLOT3D/Grid File...** ribbon tab item is used to import a PLOT3D grid file (extension `.g`, `.x`, `.xyz`, or `.grd`). See PLOT3D Files (p. 67) for details.

38.1.48. File/Import/PLOT3D/Result File...

The **File/Import/PLOT3D/Result File...** ribbon tab item s used to import a PLOT3D result file (extension `.g`, `.x`, `.xyz`, or `.grd`). See PLOT3D Files (p. 67) for details.

38.1.49. File/Import/PTC Mechanica Design...

The **File/Import/PTC Mechanica Design...** ribbon tab item is used to import a PTC Mechanica Design file (extension `.neu`), which contains analysis, model, and results data (only in binary form). See PTC Mechanica Design Files (p. 67) for details.

38.1.50. File/Import/Tecplot...

The **File/Import/Tecplot...** ribbon tab item is used to read in a Tecplot binary file. See Tecplot Files (p. 67) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.1.51. File/Import/Fluent 4 Case File...

The **File/Import/Fluent 4 Case File...** ribbon tab item is used to read in a case file created in FLUENT 4. See Fluent 4 Case Files (p. 68) for details.

38.1.52. File/Import/PreBFC File...

The **File/Import/PreBFC File...** ribbon tab item is used to read in a structured (quadrilateral or hexahedral) mesh that was created using PreBFC. See PreBFC Files (p. 68) for details.

38.1.53. File/Import/Partition/Metis...

The **File/Import/Partition/Metis...** ribbon tab item is used to partition a mesh and then read it into the parallel version of ANSYS Fluent. See Using the Partition Filter (p. 2052) for an explanation of the difference between this item and the **File/Import/Partition/Metis Zone...** ribbon tab item.

38.1.54. File/Import/Partition/Metis Zone...

The **File/Import/Partition/Metis Zone...** ribbon tab item is used to partition each cell zone in a mesh individually and then read the mesh into the parallel version of ANSYS Fluent. See Using the Partition Filter (p. 2052) for an explanation of the difference between this item and the **File/Import/Partition/Metis...** ribbon tab item.

38.1.55. File/Import/CHEMKIN Mechanism...

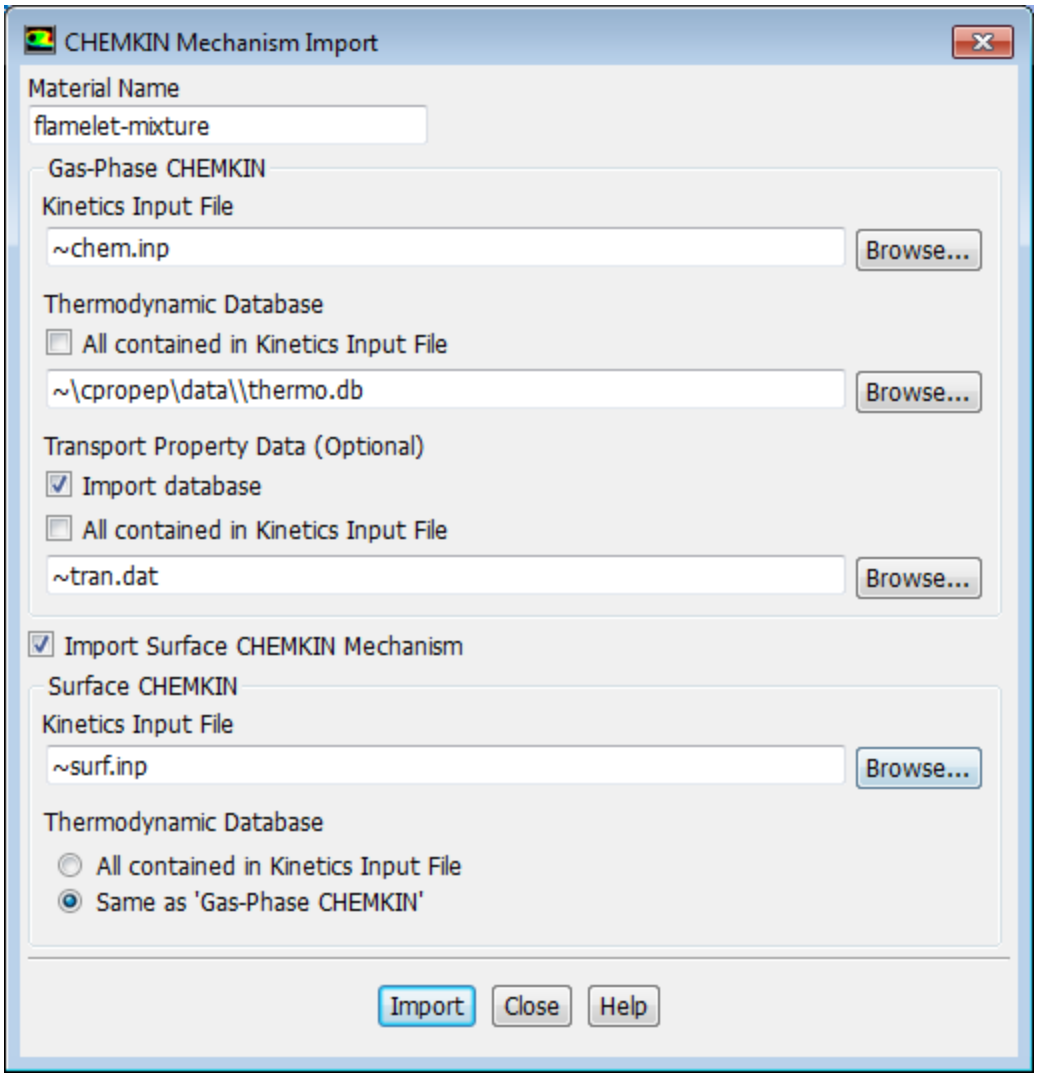
The **File/Import/CHEMKIN Mechanism...** ribbon tab item opens the CHEMKIN Mechanism Import Dialog Box (p. 2642).

38.1.55.1. CHEMKIN Mechanism Import Dialog Box

The **CHEMKIN Mechanism Import** dialog box is used to import a CHEMKIN-format chemical mechanism file into ANSYS Fluent. See Importing a Volumetric Kinetic Mechanism in CHEMKIN Format (p. 967) for details. The dialog box can be accessed either from the Species Model Dialog Box (p. 2216) or from the main menu:

File → Import → CHEMKIN Mechanism...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Material Name
specifies a name for the material.

Gas-Phase CHEMKIN
contains inputs for importing the gas-phase CHEMKIN mechanism.

Kinetics Input File
specifies the name of the gas-phase CHEMKIN-format file. If the file is not in the current working folder, include the full path to the folder where it is located.

Thermodynamic Database
specifies thermodynamic property data for species whose thermodynamic data are not specified in the kinetics input file. You can enable **All contained in Kinetics Input Files** to indicate that the kinetics input file contains the thermodynamic data for all the species.

Transport Property Data (Optional)
(if **Import database** is enabled) allows you to specify transport property data for species whose transport data are not specified in the kinetics input file. You can enable **All contained in Kinetics Input Files** to indicate that the kinetics input file contains the transport data for all the species.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Import Surface CHEMKIN Mechanism
enables the importing of the surface CHEMKIN mechanism.

Surface CHEMKIN
contains inputs for importing the surface CHEMKIN mechanism. This group box appears only when the **Import Surface CHEMKIN Mechanism** option is enabled.

Kinetics Input File
specifies the name of the surface kinetics CHEMKIN-format file.

Thermodynamic Database
specifies whether the thermodynamic data for the surface-chemistry species will be included in the surface kinetics CHEMKIN-format file (if **All contained in Kinetics Input File** is enabled) or in the thermodynamic database file specified in the **Gas-Phase CHEMKIN** group box (if **Same as 'Gas-Phase CHEMKIN'** is enabled). Note that if no thermodynamic database file is specified in the **Gas-Phase CHEMKIN** group box, the thermodynamic data for all surface-chemistry species must be included the surface kinetics input file.

Import
imports the specified Chemkin reaction mechanism and thermal files and saves the imported data in the Fluent case file. Note that carrying the original input files along with the Fluent case file is not required. ANSYS Fluent will automatically regenerate these files when needed (for example, when using the CHEMKIN-CFD solver).

38.1.56. File/Export/Solution Data...

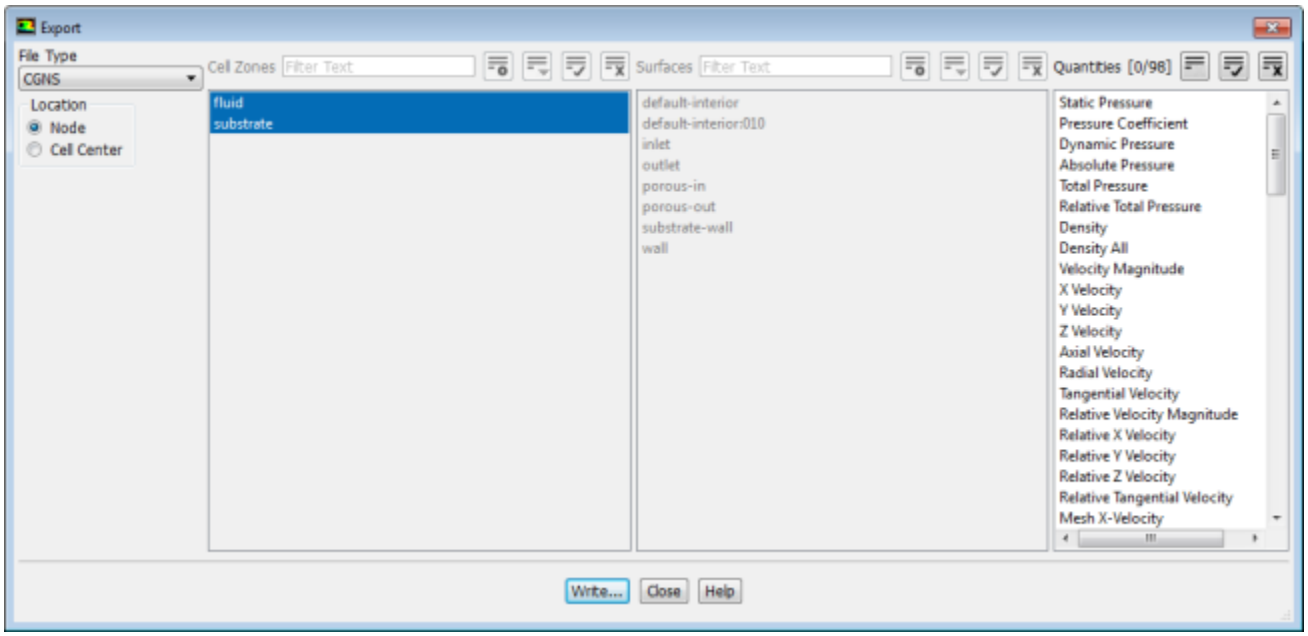
The **File/Export/Solution Data...** ribbon tab item opens the Export Dialog Box (p. 2644).

38.1.56.1. Export Dialog Box

The **Export** dialog box allows you to write data that can be read by other data visualization and post-processing tools. See Exporting Solution Data (p. 69) for a complete description of the available data formats and how to use the dialog box.

File → Export → Solution Data...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

File Type

contains a drop-down list that controls the output file format that will be written using the **Write...** button.

ABAQUS

allows you to specify the surface(s) and optional loads, based on the kind of finite element analysis selected, to be exported to an ABAQUS file (extension `.inp`).

ASCII

allows you to specify the surface(s), scalars, location in the cell from which the values of scalar functions are to be taken, and the delimiter separating the fields, to be exported to an ASCII file.

AVS

allows you to specify the scalars you want to write to be exported to an AVS file.

CFD-Post Compatible

allows you to specify the scalars you want to write, the cell zones from which the values of scalar functions are to be taken, the file format, and whether a case file is written, as part of the exporting of files that are compatible with the CFD-Post application (that is, `.cdat` and `.cst` files).

CGNS

allows you to specify the scalars you want to write and the location from which the values of scalar functions are to be taken, to be exported to a CGNS file (extension `.cgns`).

Data Explorer

allows you to specify the surface(s) and the scalars you want to write to be exported to a Data Explorer file (extension `.dx`).

EnSight Case Gold

allows you to specify the scalars you want to write, the cell zones, interior zone surfaces, and location in the cell from which the values of scalar functions are to be taken, and the file format, to be exported to an EnSight file (extension `.geo`, `.vel`, `.scl1`, or `.encas`).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FAST
allows you to specify the scalars you want to write, to be exported as a grid file (Plot3D format), a velocity file, and a scalar file. This option is available only for a triangular or tetrahedral mesh.

FAST Solution
allows you to export a single file containing density, velocity, and total energy data. This option is available only for a triangular or tetrahedral mesh.

Fieldview Unstructured
allows you to specify the scalars you want to write and the cell zones or surfaces from which the values of scalar functions are to be taken, to be exported to a FIELDVIEW binary file (extension `.fvuns`) and a regions file (extension `.fvuns.fvreg`).

Note

You must deselect all cell zones before you can select any surfaces.

I-deas Universal
allows you to specify the surface(s), scalars, and optional loads, based on the kind of finite element analysis selected, to be exported to an I-deas Universal file.

Mechanical APDL Input
allows you to specify the surface(s) and optional loads, based on the kind of finite element analysis selected, to be exported to a Mechanical APDL Input file (extension `.cdb`).

NASTRAN
allows you to specify the surface(s), scalars, and optional loads, based on the kind of finite element analysis selected, to be exported to a NASTRAN file (extension `.bdf`).

PATRAN
allows you to specify the surface(s), scalars, and optional loads, based on the kind of finite element analysis selected, to be exported to a PATRAN neutral file (extension `.out`).

RadTherm
allows you to specify the surface(s) for which you want to write data and the method of writing the heat transfer coefficient, to be exported to a PATRAN neutral file (extension `.neu`). This option is available only when the **Energy Equation** is enabled.

Tecplot
allows you to specify the surface(s) and the scalars you want to write, to be exported to a Tecplot file.

Cell Zones
specifies the cell zones for which data is to be written for a CFD-Post compatible, EnSight, or FIELDVIEW file.

Surfaces
specifies the surfaces for which data is to be written for an ABAQUS, ASCII, Data Explorer, I-deas Universal, Mechanical APDL Input, NASTRAN, PATRAN, RadTherm, or Tecplot file.

Quantities
specifies valid functions for output. The attributes of the list are modified based on the active file type. The list may be a single-selection or a multiple-selection list or it may be disabled, depending on the selected **File Type**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Structural Loads
contains optional structural loads that can be written to ABAQUS, I-deas Universal, Mechanical APDL Input, NASTRAN, and PATRAN files. This option is available only when **Structural** analysis is selected.

Force
enables force to be written as a load for a structural analysis.

Pressure
enables pressure to be written as a load for a structural analysis.

Temperature
enables temperature to be written as a load for a structural analysis. This option is available only when the **Energy Equation** is enabled.

Thermal Loads
contains optional thermal loads that can be written to ABAQUS, I-deas Universal, Mechanical APDL Input, NASTRAN, and PATRAN files. This option is available only when **Thermal** analysis is selected.

Temperature
enables temperature to be written as a load for a thermal analysis.

Heat Flux
enables heat flux to be written as a load for a thermal analysis.

Heat Trans Coeff
enables heat transfer coefficient to be written as a load for a thermal analysis.

Location (for ASCII, CGNS, and EnSight Case Gold formats)
specifies the location in the cell from which the values of scalar functions are to be taken.

Node
specifies that data values at the node points are to be exported.

Cell Center
specifies that data values from the cell centers are to be exported.

Analysis (for ABAQUS, I-deas Universal, Mechanical APDL Input, NASTRAN, and PATRAN formats
specifies the finite element analysis intended.

Structural
specifies structural analysis and allows you to select the **Structural Loads** to be written.

Thermal
specifies thermal analysis and allows you to select the **Thermal Loads** to be written.

Delimiter (for ASCII format only)
specifies the delimiter used to separate the data fields.

Comma
specifies comma as the delimiter separating the data fields.

Space
specifies space as the delimiter separating the data fields.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Transient (for EnSight Case Gold only)
contains options for exporting transient data.

Transient
enables export of transient data.

Separate Files for Each Timestep
enables writing separate files for each timestep.

Append Frequency
specifies the frequency for appending the data during the solution process.

> **Important**
>
> The **Append Frequency** option is replaced by the **Write Frequency** option when **Separate Files for Each Timestep** is enabled. You can specify the frequency for writing the separate files.

File Name
specifies the root name for the files to be saved.

Format (for EnSight Case Gold and CFD-Post Compatible)
specifies the file format.

Binary
specifies the file format as binary.

ASCII
specifies the file format as ASCII.

Heat Transfer Coefficient (for RadTherm format only)
specifies the basis for the heat transfer coefficient exported.

Flux Based
specifies the flux based method for writing the heat transfer coefficient.

Wall Function
specifies the wall function based method for writing the heat transfer coefficient.

Write Case File (for CFD-Post Compatible only)
specifies whether or not a case file is written with the `.cdat` file.

Write...
opens The Select File Dialog Box (p. 19) for writing the specified function(s) in the specified format.

38.1.57. File/Export/Particle History Data...

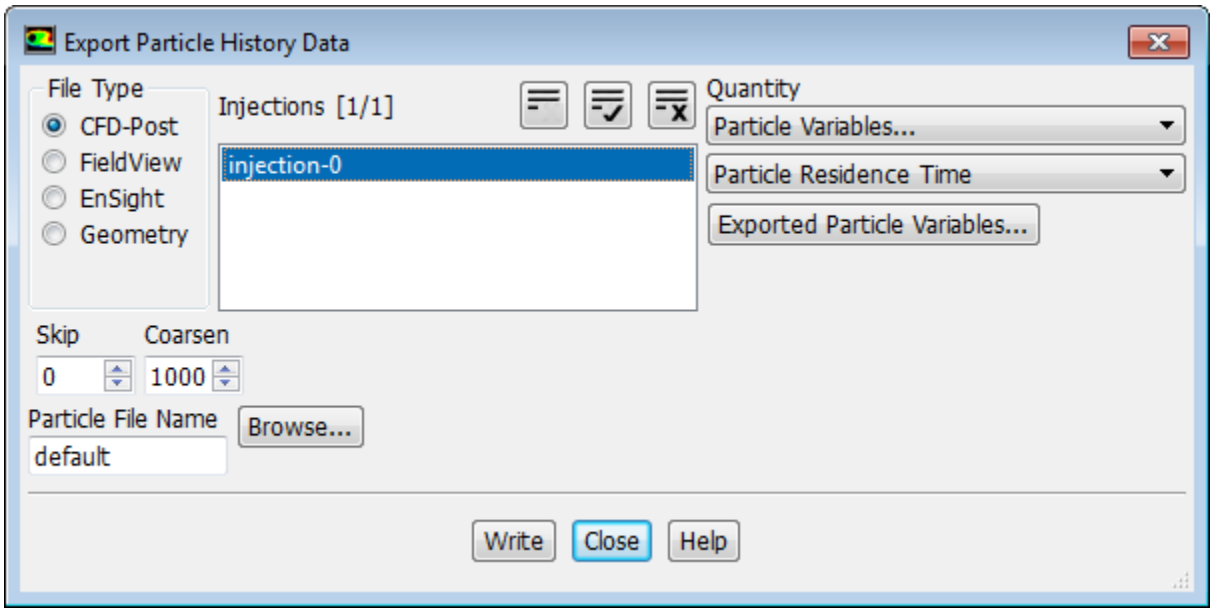
The **File/Export/Particle History Data...** ribbon tab item opens the Export Particle History Data Dialog Box (p. 2648).

38.1.57.1. Export Particle History Data Dialog Box

The **Export Particle History Data** dialog box allows you to export particle history data as your solution progresses. See Exporting Steady-State Particle History Data (p. 84) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File → Export → Particle History Data...



Controls

Type
specifies the type of the file you want to write.

CFD-Post
allows you to write the file in CFD-Post particle tracks format, which can be read in **CFD-Post**.

FieldView
allows you to write the file in **FIELDVIEW** format, which can be read in **FIELDVIEW**.

EnSight
allows you to write the file in **EnSight** format.

Geometry
allows you to write the file in `.ibl` format, which can be read in GAMBIT (not available when **Unsteady Particle Tracking** is enabled in the **Discrete Phase** dialog box, which is opened by clicking **Discrete Phase...** in the **Setting Up Physics** ribbon tab).

Injections
allows you to select the required injection from the list of predefined injections.

Quantity
contains the list of variables for which you can export the particle data.

Skip
allows you to "thin" or "sample" the number of particles that are exported.

Coarsen
reduces the exported file size by reducing the number of points that are written for a given trajectory. This is only valid for steady-state cases.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Particle File Name
allows you to specify the file name/directory for the exported data, using the **Browse...** button.

Encas File Name
is the file name you will specify if you selected **EnSight** under **Type**. Use the **Browse...** button to select the `.encas` file that was created when you exported the file with the **File/Export...** ribbon tab option.

Number of Particle Time Steps
appears when you select **EnSight** under **Type**. The specified number is the number of particle time-steps that are saved to the `.encas` file.

38.1.58. File/Export/During Calculation/Solution Data...

The **File/Export/During Calculation/Solution Data...** ribbon tab item opens the Automatic Export Dialog Box (p. 2530) (transient cases only).

38.1.59. File/Export/During Calculation/Particle History Data...

The **File/Export/During Calculation/Particle History Data...** ribbon tab item opens the Automatic Particle History Data Export Dialog Box (p. 2533) (transient cases only).

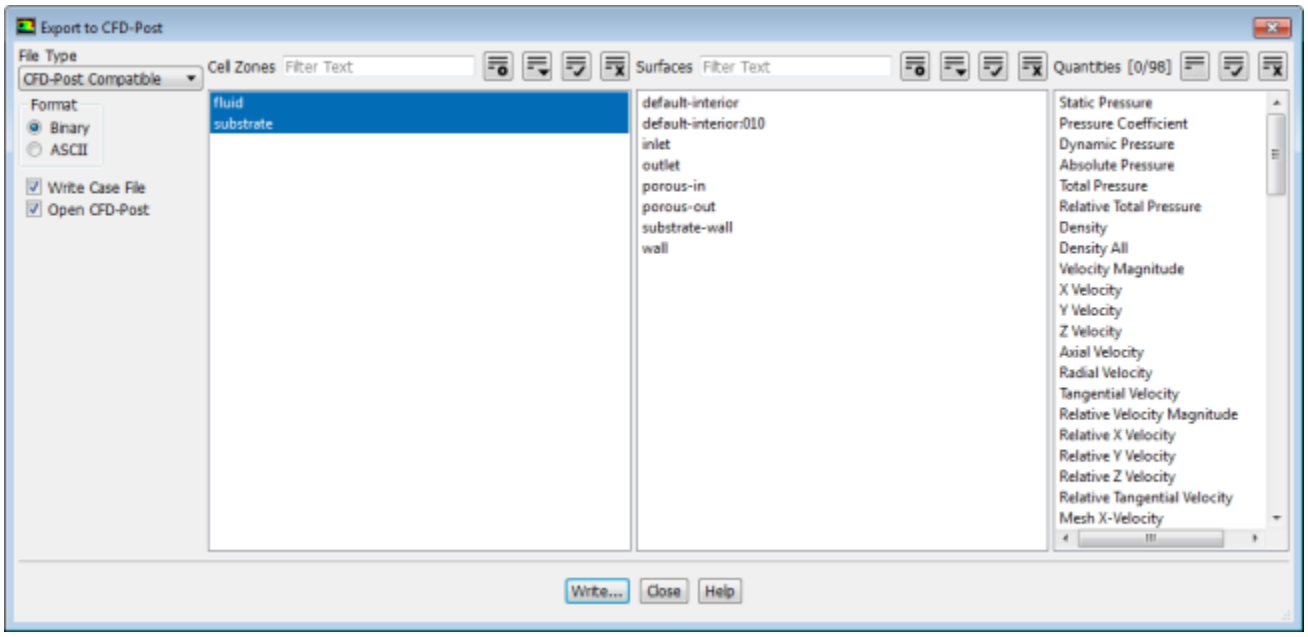
38.1.60. File/Export to CFD-Post...

The **File/Export to CFD-Post...** ribbon tab item opens the Export to CFD-Post Dialog Box (p. 2650).

38.1.60.1. Export to CFD-Post Dialog Box

The **Export to CFD-Post** dialog box allows you to select the quantities that you would like to export to CFD-Post. See Exporting to ANSYS CFD-Post (p. 91) for details.

File → Export to CFD-Post...



Controls

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select Quantities
contains a selectable list where you can choose the quantities to export.

Format
allows you to export the `.cdat` file in **Binary** or **ASCII** format.

Write Case File
specifies whether or not a case file is written with the `.cdat` file.

Open CFD-Post
specifies that after the files have been written, a CFD-Post session is opened, with the case and `.cdat` files loaded. CFD-Post will display the results.

Write...
opens the **Select File** dialog box, where you can specify the name and location of the exported files.

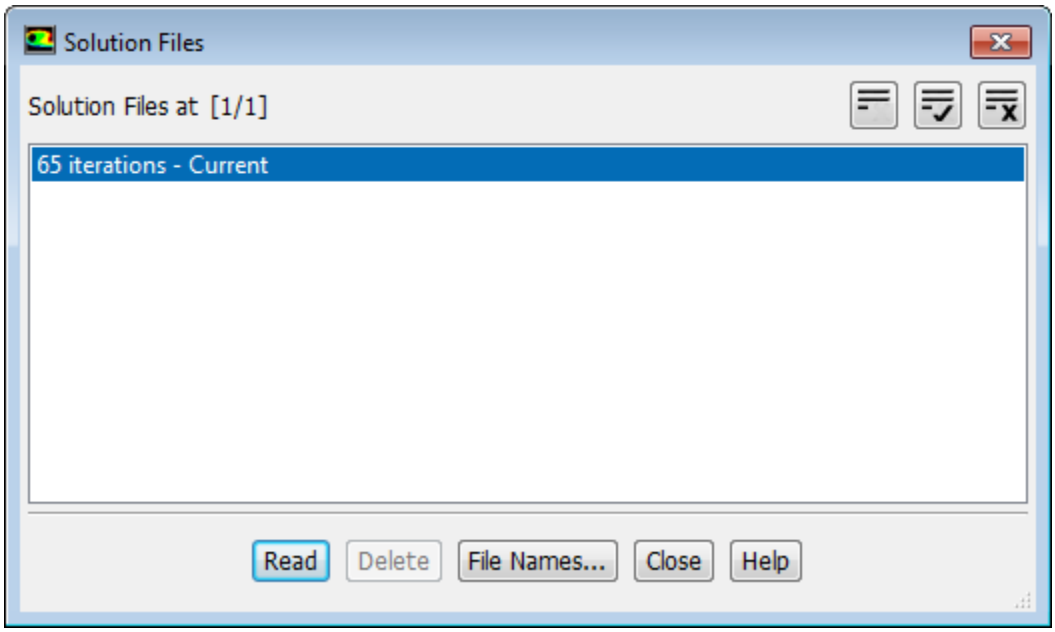
38.1.61. File/Solution Files...

The **File/Solution Files...** ribbon tab item opens the Solution Files Dialog Box (p. 2651).

38.1.61.1. Solution Files Dialog Box

The **Solution Files** dialog box allows you to manage the files that were created through the Autosave Dialog Box (p. 2527). See Managing Solution Files (p. 93) for details.

File → Solution Files...



Controls

Solution Files at
contains a selectable list where you can choose the files to read or delete.

Read
makes the selected file the current file. Note that if more than one file is selected, the **Read** button is disabled. When an earlier solution is made current, the solution files that were generated for a later iteration/time-step will be removed from this list when the calculation continues.

Delete
removes the selected solution files.

File Names...
allows you to obtain information about the solution files and the path of the associated files.

Recover Missing Solution...
Recovers the latest case and data files from the system folder (<*project name*>\dp0\FFF\Fluent\ for analysis system and <*project name*>\dp0\ FLU \Fluent\ for component system) and list them in the **Solution Files at** list. (This option is available only in Fluent used under Workbench.)

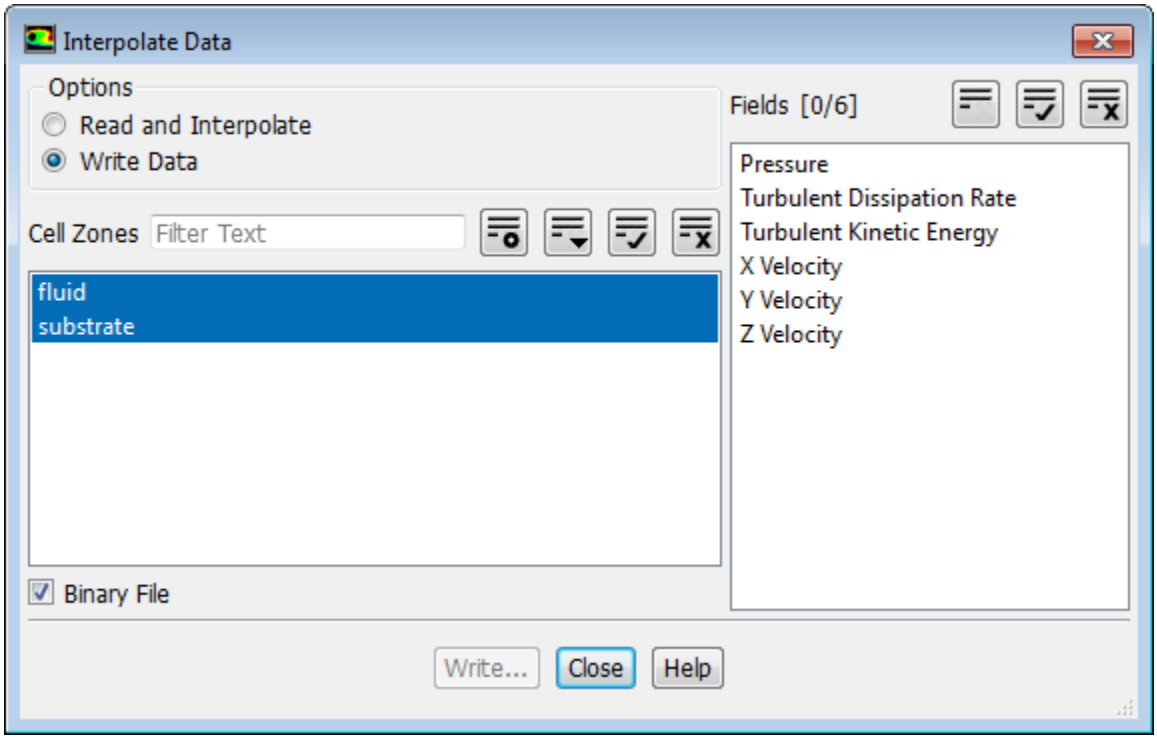
38.1.62. File/Interpolate...

The **File/Interpolate...** ribbon tab item opens the Interpolate Data Dialog Box (p. 2652).

38.1.62.1. Interpolate Data Dialog Box

The **Interpolate Data** dialog box allows you to interpolate solution data from one mesh to another. See Mesh-to-Mesh Solution Interpolation (p. 94) for details.

File → Interpolate...



Controls

Options
contains the interpolation options.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Read and Interpolate
allows you to read and interpolate solution data onto the current mesh.

Write Data
allows you to write an interpolation file for the solution data to be interpolated onto another mesh.

Cell Zones
is a list of cell zones that can be selected.

Binary File
allows you to write binary interpolation files. This option is available only when **Write Data** is selected under **Options**.

Fields
is a list of all available data fields that can be selected. This list is available only when **Write Data** is selected under **Options**.

Read...
opens The Select File Dialog Box (p. 19), in which you can specify the file to be read. This button is available only when the **Read and Interpolate** option is selected.

Write...
opens The Select File Dialog Box (p. 19), in which you can specify a name for the file to be saved and then save the file. This button replaces the **Read...** button when the **Write Data** option is selected.

38.1.63. File/FSI Mapping/Volume...

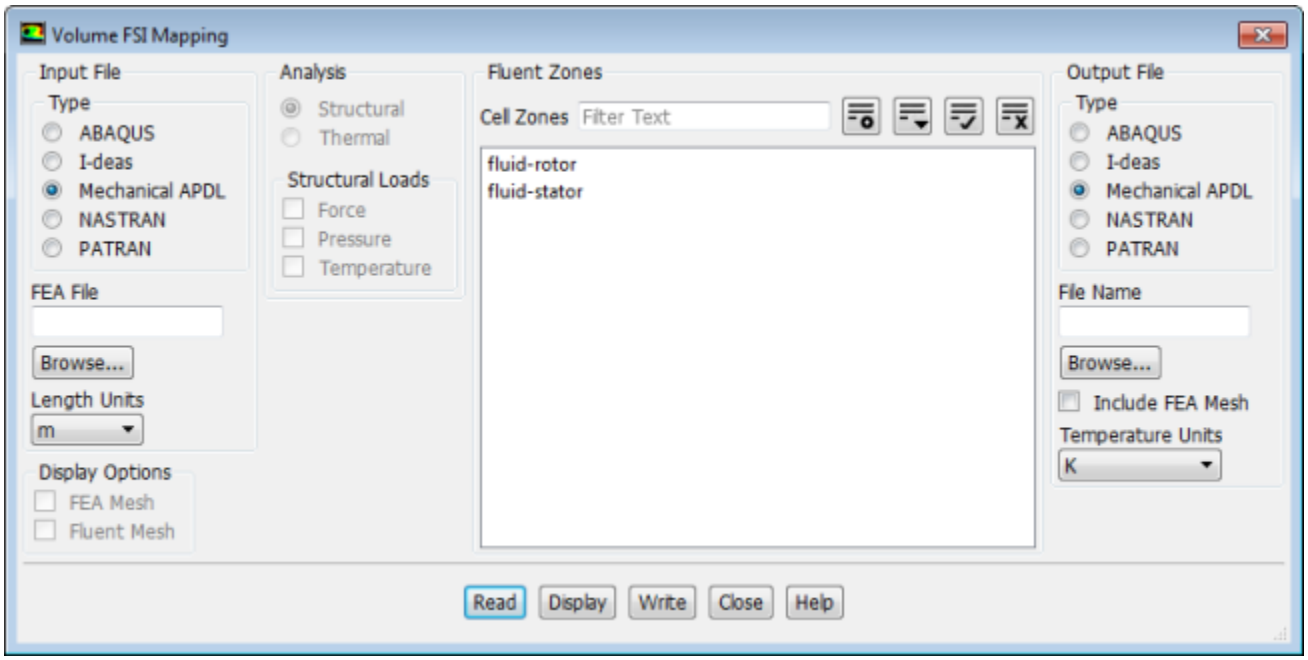
The **File/FSI Mapping/Volume...** ribbon tab item opens the Volume FSI Mapping Dialog Box (p. 2653).

38.1.63.1. Volume FSI Mapping Dialog Box

The **Volume FSI Mapping** dialog box allows you to map cell data for a given geometry from an ANSYS Fluent file onto a file with a different mesh and format. See Mapping Data for Fluid-Structure Interaction (FSI) Applications (p. 97) for a complete description of how to use the dialog box.

File → FSI Mapping → Volume...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Input File

contains parameters related to the input file.

Type

contains the options for the format of the input mesh file.

ABAQUS

specifies that an ABAQUS file will be used as the input mesh file (extension `.inp`).

I-deas

specifies that an I-deas file will be used as the input mesh file (extension `.unv`).

Mechanical APDL

specifies that a Mechanical APDL file will be used as the input mesh file (extension `.cdb` or `.neu`).

NASTRAN

specifies that a NASTRAN file will be used as the input mesh file (extension `.bdf`).

PATRAN

specifies that a PATRAN file will be used as the input mesh file (extension `.neu`, `.out`, or `.pat`).

FEA File

specifies the name of the input mesh file (see the **Input File Type** descriptions for associated file extensions). If the file is not in the current working folder, include the full path to the folder where it is located.

Browse...

opens The Select File Dialog Box (p. 19), which you can use to specify the input mesh file instead of entering it in the **Input File** text-entry box.

Length Units

specifies the unit of length used in the input file.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Display Options
allows you to choose either the **FEA Mesh** or the **ANSYS Fluent Mesh** (or both).

Analysis
contains the options for the kind of data to be mapped.

Structural
specifies that data fields relevant for a structural analysis will be mapped.

Thermal
specifies that data fields relevant for a thermal analysis will be mapped.

Structural Loads
consists of a list of available loads, including **Force**, **Pressure**, and **Temperature**.

Fluent Zones
contains a list of available cell zones from the current ANSYS Fluent file from which cell data will be mapped.

Output File
contains the options for the format of the file that will be created from the input mesh file and the mapped data.

Type
contains the options for the format of the output mesh file.

ABAQUS
specifies that an ABAQUS file will be used as the output mesh file (extension `.inp`).

I-deas
specifies that an I-deas file will be used as the output mesh file (extension `.unv`).

Mechanical APDL
specifies that a Mechanical APDL file will be used as the output mesh file (extension `.cdb`).

NASTRAN
specifies that a NASTRAN file will be used as the output mesh file (extension `.bdf`).

PATRAN
specifies that a PATRAN file will be used as the output mesh file (extension `.out`).

File Name
specifies the name of the input mesh file (see the **Output File** descriptions for associated file extensions). If the file is not in the current working folder, include the full path to the folder where it is located.

Browse...
opens The Select File Dialog Box (p. 19), which you can use to specify the input mesh file instead of entering it in the **Input File** text-entry box.

Include FEA Mesh
includes additional FEA information like node/element information in the exported output file.

Temperature Units
specifies the unit of temperature when mapping temperature for a structural analysis or any variable for a thermal analysis.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Read
reads the input file into memory.

Display
displays the selected mesh in the graphics window.

Write
writes an output file in which the selected ANSYS Fluent cell data is mapped onto the input mesh file.

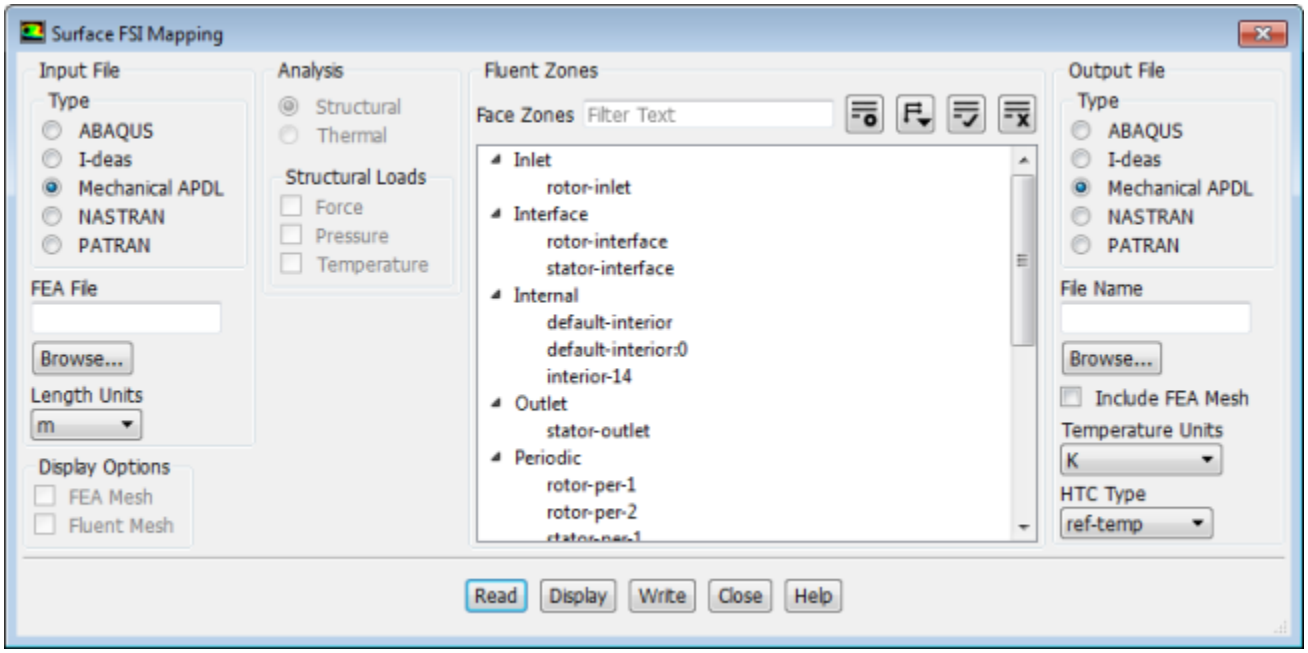
38.1.64. File/FSI Mapping/Surface...

The **File/FSI Mapping/Surface...** ribbon tab item opens the Surface FSI Mapping Dialog Box (p. 2656).

38.1.64.1. Surface FSI Mapping Dialog Box

The **Surface FSI Mapping** dialog box allows you to map face data for a given geometry from an ANSYS Fluent file to a file with a different mesh and format. See Mapping Data for Fluid-Structure Interaction (FSI) Applications (p. 97) for a complete description of how to use the dialog box.

File → FSI Mapping → Surface...



Controls

Input File
contains parameters related to the input file.

Type
contains the options for the format of the input mesh file.

ABAQUS
specifies that an ABAQUS file will be used as the input mesh file (extension `.inp`).

I-deas
specifies that an I-deas file will be used as the input mesh file (extension `.unv`).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mechanical APDL
specifies that a Mechanical APDL file will be used as the input mesh file (extension `.cdb` or `.neu`).

NASTRAN
specifies that a NASTRAN file will be used as the input mesh file (extension `.bdf`).

PATRAN
specifies that a PATRAN file will be used as the input mesh file (extension `.neu`, `.out`, or `.pat`).

FEA File
specifies the name of the input mesh file (see the **Input File Type** descriptions for associated file extensions). If the file is not in the current working folder, include the full path to the folder where it is located.

Browse...
opens The Select File Dialog Box (p. 19), which you can use to specify the input mesh file instead of entering it in the **Input File** text-entry box.

Length Units
specifies the unit of length used in the input file.

Display Options
allows you to choose either the **FEA Mesh** or the **ANSYS Fluent Mesh** (or both).

Analysis
contains the options for the kind of data to be mapped.

Structural
specifies that data fields relevant for a structural analysis will be mapped.

Thermal
specifies that data fields relevant for a thermal analysis will be mapped.

Structural Loads
consists of a list of available loads, including **Force**, **Pressure**, and **Temperature**.

Fluent Zones
contains a list of available face zones from the current ANSYS Fluent file from which cell data will be mapped.

Output File
contains the options for the format of the file that will be created from the input mesh file and the mapped data.

Type
contains the options for the format of the output mesh file.

ABAQUS
specifies that an ABAQUS file will be used as the output mesh file (extension `.inp`).

I-deas
specifies that an I-deas file will be used as the output mesh file (extension `.unv`).

Mechanical APDL
specifies that a Mechanical APDL file will be used as the output mesh file (extension `.cdb`).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NASTRAN
specifies that a NASTRAN file will be used as the output mesh file (extension `.bdf`).

PATRAN
specifies that a PATRAN file will be used as the output mesh file (extension `.out`).

File Name
specifies the name of the input mesh file (see the **Output File** descriptions for associated file extensions). If the file is not in the current working folder, include the full path to the folder where it is located.

Browse...
opens The Select File Dialog Box (p. 19), which you can use to specify the input mesh file instead of entering it in the **Input File** text-entry box.

Include FEA Mesh
includes additional FEA information like node/element information in the exported output file.

Temperature Units
specifies the unit of temperature when mapping temperature for a structural analysis or any variable for a thermal analysis.

HTC Type
specifies the means of calculating the heat transfer coefficient.

ref-temp
uses a temperature equal to the reference temperature to calculate the heat transfer coefficient.

cell-temp
uses a temperature equal to the temperature of the cell adjacent to the face to calculate the heat transfer coefficient.

wall-func-htc
calculates h_{eff} using Equation 33.55 (p. 2001). Note that this option has the same definition as the field variable **Wall Func. Heat Tran. Coef.**, as described in Alphabetical Listing of Field Variables and Their Definitions (p. 1957).

Read
reads the input file into memory.

Display
displays the selected mesh in the graphics window.

Write
writes an output file in which the ANSYS Fluent data has been mapped to the mesh of the input file.

38.1.65. File/Save Picture...

The **File/Save Picture...** ribbon tab item opens the Save Picture Dialog Box (p. 2568).

38.1.66. File/Data File Quantities...

The **File/Data File Quantities...** ribbon tab item opens the Data File Quantities Dialog Box (p. 2529).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

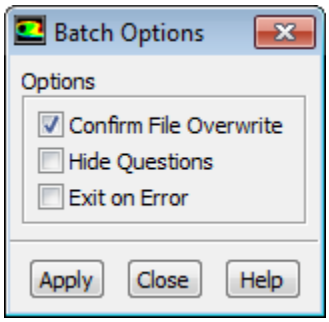
38.1.67. File/Batch Options...

The **File/Batch Options...** ribbon tab item opens the Batch Options Dialog Box (p. 2659).

38.1.67.1. Batch Options Dialog Box

The **Batch Options** dialog box allows you to select options to suppress interactive dialog boxes from ANSYS Fluent while running a case in batch mode. See Batch Execution Options in the Getting Started Guide for details.

File → Batch Options...



Controls

Confirm File Overwrite
determines whether ANSYS Fluent confirms a file overwrite. This option is enabled by default.

Hide Questions
allows you to hide **Question** dialog boxes. This option is disabled by default.

Exit on Error
allows you to automatically exit from batch mode when an error occurs. This option is disabled by default.

When run in batch mode through the command prompt or a journal file with **Exit on Error** enabled, Fluent will exit under the following circumstances:

- Normal run termination upon reaching the end of a journal (return value `0`)
- Error returned during scripted text command execution (return value `1`)
- Unexpected input (wrong type) to text command (return value `1`)
- Licensing error (return value 2)

If an invalid text command is entered, Fluent will not exit, but proceed to the next text input.

Note that in Windows you must start Fluent with the `-wait` command line option.

38.1.68. File/Exit

The **File/Exit** ribbon tab item is used to exit from the current solver session.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.2. Dialog Boxes Available from the Ribbon

The dialog boxes described in this section can be accessed through the ribbon, from the tabs other than the **File** ribbon tab.

38.2.1. 1D Simulation Library Dialog Box
38.2.2. Activate Cell Zones Dialog Box
38.2.3. Adaption Display Options Dialog Box
38.2.4. Adjacency Dialog Box
38.2.5. Animation Definition Dialog Box
38.2.6. Animation Recording Options Dialog Box
38.2.7. Anisotropic Adaption Dialog Box
38.2.8. Auto Create Options Dialog Box
38.2.9. Auto Partition Mesh Dialog Box
38.2.10. Boundary Adaption Dialog Box
38.2.11. Compiled UDFs Dialog Box
38.2.12. Contours Dialog Box
38.2.13. Convergence Conditions Dialog Box
38.2.14. Create/Edit Mesh Interfaces Dialog Box
38.2.15. Custom Field Function Calculator Dialog Box
38.2.16. Custom Laws Dialog Box
38.2.17. Deactivate Cell Zones Dialog Box
38.2.18. Define Control Points Dialog Box
38.2.19. Delete Cell Zones Dialog Box
38.2.20. DPM Report Definition Dialog Box
38.2.21. DPM Source Report Definition
38.2.22. Drag Report Definition Dialog Box
38.2.23. DTRM Graphics Dialog Box
38.2.24. DTRM Rays Dialog Box
38.2.25. Edit Mesh Interfaces Dialog Box
38.2.26. Edit Report File Dialog Box
38.2.27. Edit Report Plot Dialog Box
38.2.28. Execute on Demand Dialog Box
38.2.29. Expression Report Definition Dialog Box
38.2.30. Field Function Definitions Dialog Box
38.2.31. Flux Report Definition Dialog Box
38.2.32. Fuse Face Zones Dialog Box
38.2.33. Geometry Based Adaption Controls Dialog Box
38.2.34. Geometry Based Adaption Dialog Box
38.2.35. Gradient Adaption Dialog Box
38.2.36. Import Particle Data Dialog Box
38.2.37. Imprint Surface Dialog Box
38.2.38. Improve Mesh Dialog Box
38.2.39. Injections Dialog Box
38.2.40. Input Summary Dialog Box
38.2.41. Interface Options Dialog Box
38.2.42. Interpreted UDFs Dialog Box
38.2.43. Iso-Clip Dialog Box
38.2.44. Iso-Surface Dialog Box
38.2.45. Iso-Value Adaption Dialog Box
38.2.46. Lift Report Definition Dialog Box
38.2.47. Line/Rake Surface Dialog Box
38.2.48. Manage Adaption Registers Dialog Box

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.2.49. Merge Zones Dialog Box
38.2.50. Mesh Adaption Controls Dialog Box
38.2.51. Mesh Interfaces Dialog Box
38.2.52. Mesh Morpher/Optimizer Dialog Box
38.2.53. MiniVAS Settings Dialog Box
38.2.54. Mixing Planes Dialog Box
38.2.55. Moment Report Definition Dialog Box
38.2.56. Motion Settings Dialog Box
38.2.57. New Report File Dialog Box
38.2.58. New Report Plot Dialog Box
38.2.59. Objective Function Definition Dialog Box
38.2.60. Optimization History Monitor Dialog Box
38.2.61. Parallel Connectivity Dialog Box
38.2.62. Parameter Bounds Dialog Box
38.2.63. Particle Tracks Dialog Box
38.2.64. Partition Surface Dialog Box
38.2.65. Partitioning and Load Balancing Dialog Box
38.2.66. Pathlines Dialog Box
38.2.67. Picture Options Dialog Box
38.2.68. Plane Surface Dialog Box
38.2.69. Point Surface Dialog Box
38.2.70. Quadric Surface Dialog Box
38.2.71. Region Adaption Dialog Box
38.2.72. Replace Cell Zone Dialog Box
38.2.73. Report Definitions Dialog Box
38.2.74. Report File Definitions Dialog Box
38.2.75. Report Plot Definitions Dialog Box
38.2.76. Residual Monitors Dialog Box
38.2.77. Rotate Mesh Dialog Box
38.2.78. S2S Information Dialog Box
38.2.79. Separate Cell Zones Dialog Box
38.2.80. Separate Face Zones Dialog Box
38.2.81. Set Injection Properties Dialog Box
38.2.82. Set Multiple Injection Properties Dialog Box
38.2.83. Shell Conduction Layers Dialog Box
38.2.84. Shell Conduction Manager Dialog Box
38.2.85. Surface Meshes Dialog Box
38.2.86. Surface Report Definition Dialog Box
38.2.87. Surfaces Dialog Box
38.2.88. Thread Control Dialog Box
38.2.89. Transform Surface Dialog Box
38.2.90. Translate Mesh Dialog Box
38.2.91. Turbo 2D Contours Dialog Box
38.2.92. Turbo Averaged Contours Dialog Box
38.2.93. Turbo Averaged XY Plot Dialog Box
38.2.94. Turbo Options Dialog Box
38.2.95. Turbo Report Dialog Box
38.2.96. Turbo Topology Dialog Box
38.2.97. UDF Library Manager Dialog Box
38.2.98. User-Defined Fan Model Dialog Box
38.2.99. User-Defined Function Hooks Dialog Box
38.2.100. User-Defined Memory Dialog Box

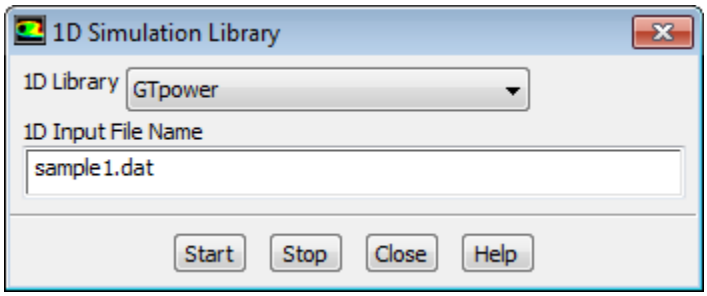
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.2.101. User Defined Report Definition Dialog Box
38.2.102. User-Defined Scalars Dialog Box
38.2.103. Vectors Dialog Box
38.2.104. Video Control Dialog Box
38.2.105. V-LAN Settings Dialog Box
38.2.106. Volume Adaption Dialog Box
38.2.107. Volume Report Definition Dialog Box
38.2.108. Warning Dialog Box
38.2.109. Yplus/Ystar Adaption Dialog Box
38.2.110. Zone Surface Dialog Box

38.2.1. 1D Simulation Library Dialog Box

The **1D Simulation Library** dialog box allows you to set parameters related to coupling between ANSYS Fluent and GT-POWER or WAVE. See Coupling Boundary Conditions with GT-POWER (p. 451) or Coupling Boundary Conditions with WAVE (p. 456) for details about the items below.

User Defined → **Model Specific** → **1D Coupling...**



Controls

1D Library
specifies the type of library to be used. (Currently only **GTpower** and **WAVE** are available.)

1D Input File Name
specifies the name of the GT-POWER or WAVE input file.

Start
starts up GT-POWER or WAVE and generates ANSYS Fluent user-defined functions for each boundary in the input file.

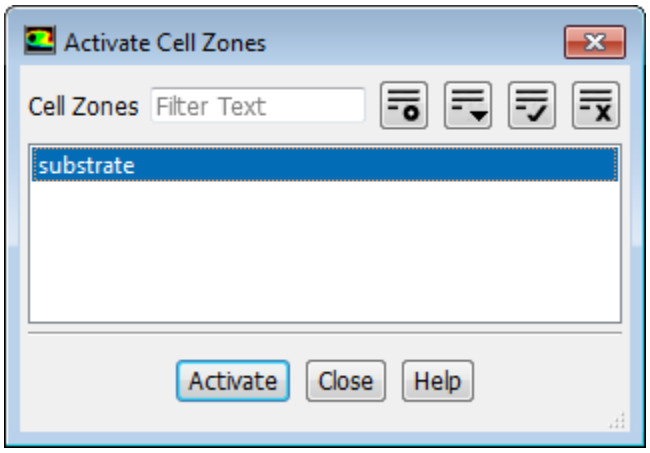
Stop
unlinks the shared library.

38.2.2. Activate Cell Zones Dialog Box

The **Activate Cell Zones** dialog box allows you to activate a single cell zone or multiple zones. See Activating Zones (p. 221) for details.

Setting Up Domain → **Zones** → **Activate...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

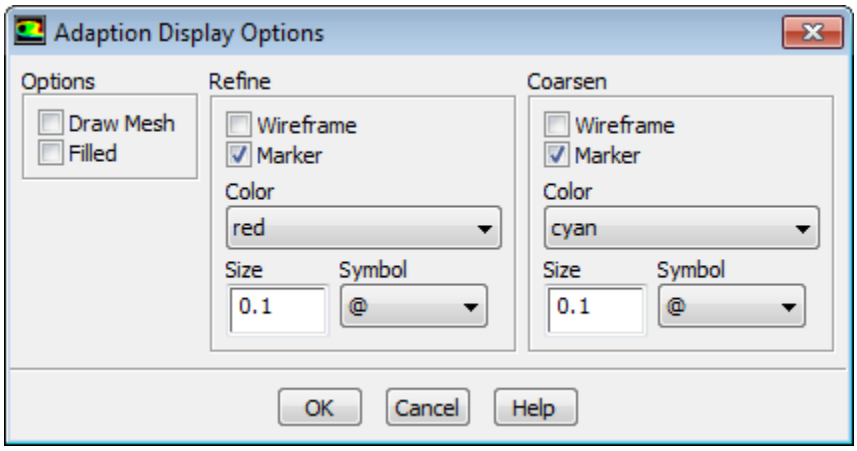
Cell Zones
contains a list of cell zones from which you can select the zone to be activated.

Activate
activates the selected cell zones.

38.2.3. Adaption Display Options Dialog Box

The **Adaption Display Options** dialog box allows you to customize the display of adaption or mask registers. See Adaption Display Options (p. 1704) for details about the items below.

Setting Up Domain → Adapt → More → Display Options...



Controls

Options
contains check buttons that control the drawing of the mesh and the type of graphical tool used to display flagged cells.

Draw Mesh
toggles the ability to draw the mesh with the adaption display. This command opens the Mesh Display Dialog Box (p. 2163), which allows you to select the desired surface or zone meshes to be displayed with the markings.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Filled
toggles the solid shading of the cell wireframe.

Refine
contains options related to the display of cells marked for refinement.

Wireframe
toggles the display of the cell wireframe for cells flagged for refinement.

Marker
toggles the display of the cell marker for cells flagged for refinement.

Color
is a drop-down list of colors for the wireframe or marker for the cells marked for refinement.

Size
is a real number entry for the size of the refine cell marker. A symbol of size 1.0 is 3.0% of the height of the display screen.

Symbol
is a drop-down list of symbols that can be used for the refine cell marker.

Coarsen
contains options related to the display of cells marked for refinement.

Wireframe
toggles the display of the cell wireframe for cells flagged for coarsening.

Marker
toggles the display of the cell marker for cells flagged for coarsening.

Color
is a drop-down list of colors for the wireframe or marker for the cells marked for coarsening.

Size
is a real number entry for the size of the coarsen cell marker. A symbol of size 1.0 is 3.0% of the height of the display screen.

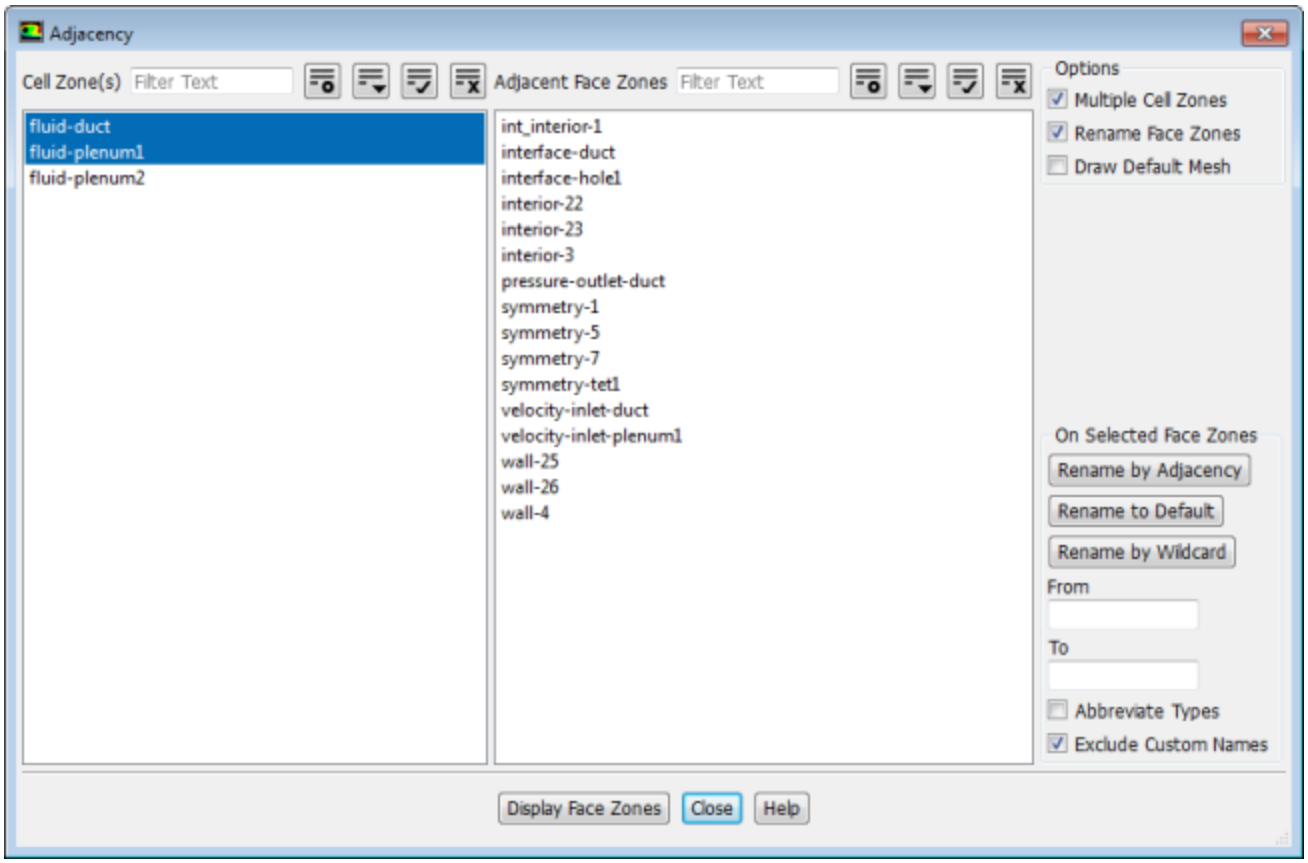
Symbol
is a drop-down list of symbols that can be used for the coarsen cell marker.

38.2.4. Adjacency Dialog Box

The **Adjacency** dialog box allows you to identify, display, and rename face zones based on their adjacency to a selected cell zone. See Managing Adjacent Zones (p. 224) for details.

Setting Up Domain → Zones → Adjacency...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Cell Zone(s)
contains a list of the cell zones for which you can find adjacent face zones.

Adjacent Face Zones
contains a list of the face zones that are adjacent to the selected cell zones.

Options

Multiple Cell Zones
allows you to select multiple cell zones at once. The zones listed in **Adjacent Face Zones** will be the union of all zones adjacent to the selected cell zones.

Rename Face Zones
allows you to rename selected face zones based on adjacency or zone type.

Draw Default Mesh
brings up the Mesh Display Dialog Box (p. 2163) where you can specify zones of the mesh to be permanently displayed as you display or hide face zones from within the **Adjacency** dialog box. This can be helpful in cases where the mesh is very complex. If this option is not enabled, only the zones selected in **Adjacent Face Zones** will be displayed when you click **Display Face Zones**.

On Selected Face Zones
contains controls for renaming face zones that are selected in **Adjacent Face Zones**. For details on the renaming methods, refer to Renaming Zones Using the Adjacency Dialog Box (p. 225).

Rename by Adjacency
renames the selected face zones incorporating the name of the adjacent cell zone and the face zone type.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rename to Default
renames the selected face zones with default names based on the face zone type and, if necessary to avoid duplicate names, the zone id.

Rename by Wildcard
renames the selected face zones based on a pattern match string in the **From** field and a replacement string in the **To** field.

Abbreviate Types
uses abbreviations for the zone types when renaming rather than the full zone type text.

Exclude Custom Names
excludes from renaming any zones that do not match a recognized naming pattern. This can be useful to prevent inadvertently replacing a meaningful name.

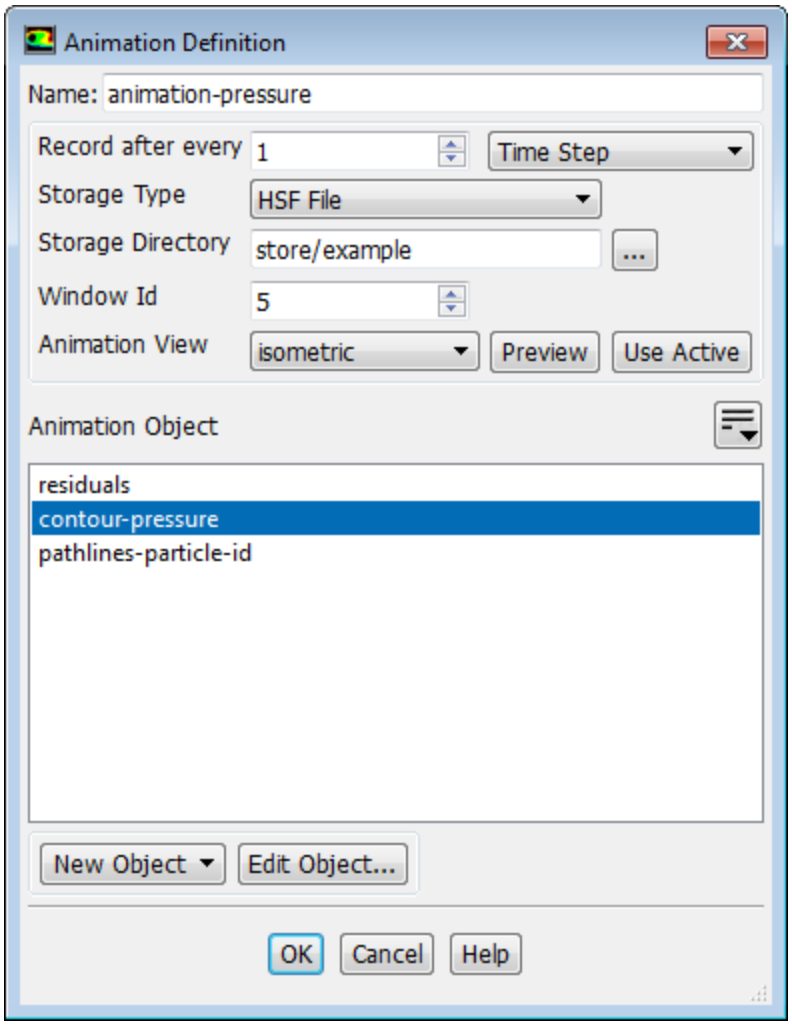
Displace Face Zones
displays the face zones selected in **Adjacent Face Zones**.

38.2.5. Animation Definition Dialog Box

The **Animation Definition** dialog box allows you to specify a graphics object that will be captured during the solution calculation so that it may be played back as an animation during postprocessing.

Solving → Activities → Create → Solution Animations...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name
name of the animation definition.

Record after every
specifies the frequency that images are captured during the calculation. You can select whether the frequency is **Iteration** or **Time Step** (transient only) from the drop-down.

Storage Type
drop-down list with the available formats for storing the animation images.

In Memory
the image files associated with the animation will lost when you exit Fluent.

PPM Image
Fluent will save the animation images in PPM Image format (`.cxa`), which is 2D.

HSF File
Fluent will save the animation images in HSF file format (`.hsf`), which is 3D.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Storage Directory
shows the location where the animation images will be stored if the **Storage Type** is set to **PPM Image** or **HSF File**.

Window ID
indicates the window where the solution animation images will be displayed when the solution is calculating.

Animation View
specifies the animation object orientation that will be used when the animation images are capture during the calculation.

views drop-down
list with all of the defined views.

Preview
displays the selected animation object in the graphics window specified by **Window ID**.

Use Active
saves the current view in the active graphics window and automatically selects it for use in the animation definition.

Animation Object
lists the defined graphics objects that may be selected for animation.

New Object
drop-down list allowing you to create new graphics objects for animating.

- **Mesh...** opens the Mesh Display Dialog Box (p. 2163).
- **Contours...** opens the Contours Dialog Box (p. 2676).
- **Vectors...** opens the Vectors Dialog Box (p. 2831).
- **Pathlines...** opens the Pathlines Dialog Box (p. 2771).
- **Particle Tracks...** opens the Particle Tracks Dialog Box (p. 2761).
- **Scene...** opens Figure 31.26: The Scene Dialog Box (p. 1781).
- **XY Plot...** opens the Solution XY Plot Dialog Box (p. 2595).
- **Report Plot...** opens the New Report Plot Dialog Box (p. 2755).

Edit Object...
allows you to edit the selected graphics object.

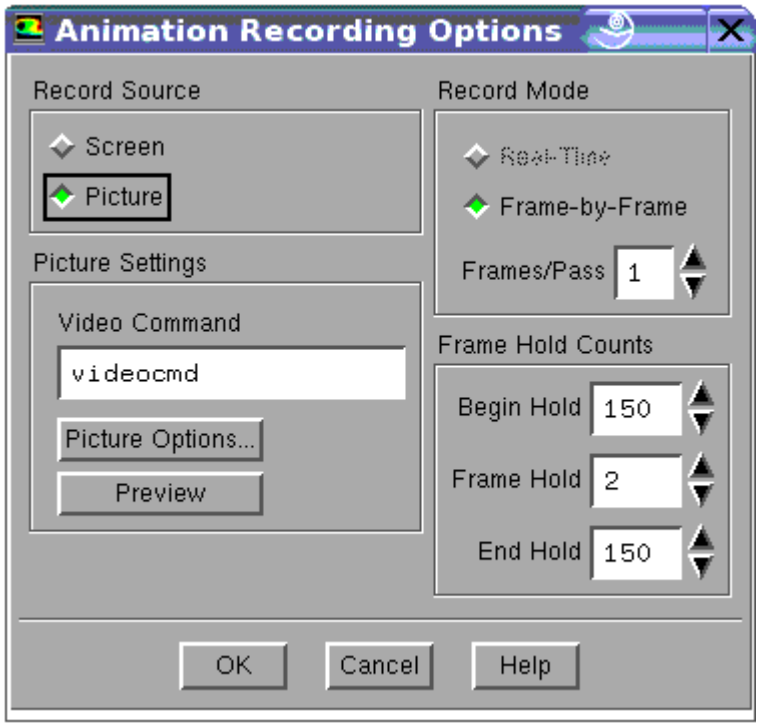
OK
creates the animation definition and/or saves settings changes and closes the dialog box.

38.2.6. Animation Recording Options Dialog Box

The **Animation Recording Options** dialog box allows you to set specific options for recording animations to video. An animation is created using the Animate Dialog Box (p. 2567).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Animation Recording Options** dialog box is opened from the Video Control Dialog Box (p. 2833).



Controls

Record Source
identifies the video source used during a recording session.

Screen
informs ANSYS Fluent that the computer's video hardware will send all or a portion of the computer screen as a video signal to the VTR. This option assumes your computer's video system includes a scan converter and associated software that converts the computer RGB signal to a video signal. You are responsible for setting up the scan converter and sending the video signal to the VTR.

Picture
instructs ANSYS Fluent to create a picture of each frame of animation and send the picture file to the computer's video hardware using a system command. This option assumes that your computer's video system includes a frame buffer that can store an image and send it as a video signal to the video recording system.

Picture Settings
is used in conjunction with the **Picture** record source option.

Video Command
identifies the shell script command file used to send a picture file to the video frame buffer. The default setting is `videocmd`, which is a shell script that is included in your ANSYS Fluent distribution. It is located in *path* `/ansys_inc/v181/fluent/bin`, where *path* is the directory in which you have placed the release directory. This shell script will run your system's command to send an image file to the video frame buffer. The script `videocmd` is set up to call the SGI system command `memtovid`. If you have a different system, you must copy the shell script `videocmd` to a new file and modify it to perform the proper task on your system (see the comments in `videocmd` for details).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Picture Options...
displays the Save Picture Dialog Box (p. 2568), which allows you to set up the picture format supported by your video frame buffer. If you choose to perform a window dump to create the picture file, the default window dump command used will also be `videocmd`. You can change this setting to use your own command. After setting the picture options, click **Apply** instead of **Save...** in the Save Picture Dialog Box (p. 2568) to effect the change.

Preview
will send a test picture of the current graphics picture to the video frame buffer. This can be used to check the image on your video recording system's monitor to see if the image size and color levels are acceptable.

Record Mode
specifies the method used to record the animation.

Real-Time
can be used if the animation playback speed is fast enough to provide a reasonably smooth animation in real-time. This is only available if the selected record source is **Screen**. In this mode, ANSYS Fluent will simply turn VTR recording on, play the animation, then stop the recording.

Frame-By-Frame
is used to produce a higher-quality video animation by recording one frame at a time. For each animation frame, this method will 1) play the frame on the screen (and generate the picture file, if needed), 2) preroll the VTR, and 3) record the frame. If the animation has 50 frames, this procedure is repeated 50 times, that is, 50 record passes are made. This is the recommended method, because the real-time playback of the animation will usually be too slow and choppy.

Frames/Pass
is used in conjunction with **Frame-By-Frame** recording to try and speed up this process, if possible. It specifies the number of animation frames recorded to tape per record pass. If the animation is long enough (200 frames or more), you can try setting this value to 2 or higher. For example, if you set this value to 2 for a 202-frame animation, it will record animation frame 1 during the first pass, frames 2 and 102 during the second pass, frames 3 and 103 during the third pass, and so on. This is possible only if the animation frames can be rendered in time to be inserted onto the tape during a record pass, so use this setting with caution.

Frame Hold Counts
specifies the number of video frames to hold an animation frame during recording. The video standard NTSC has a frame rate of 30 frames/sec. At that rate, a 150-frame animation will take only 5 seconds to play. To stretch out the animation, you can record the same animation frame over 2 or more video frames.

Begin Hold
specifies the number of video frames to hold the first animation frame. It helps to hold the first frame for about 5 seconds (150 video frames) so that the viewer can get accustomed to the picture before the animation begins.

Frame Hold
specifies the number of video frames to hold each animation frame, other than the first and last. To slow down your recorded animation, try setting this value to 2 or 3.

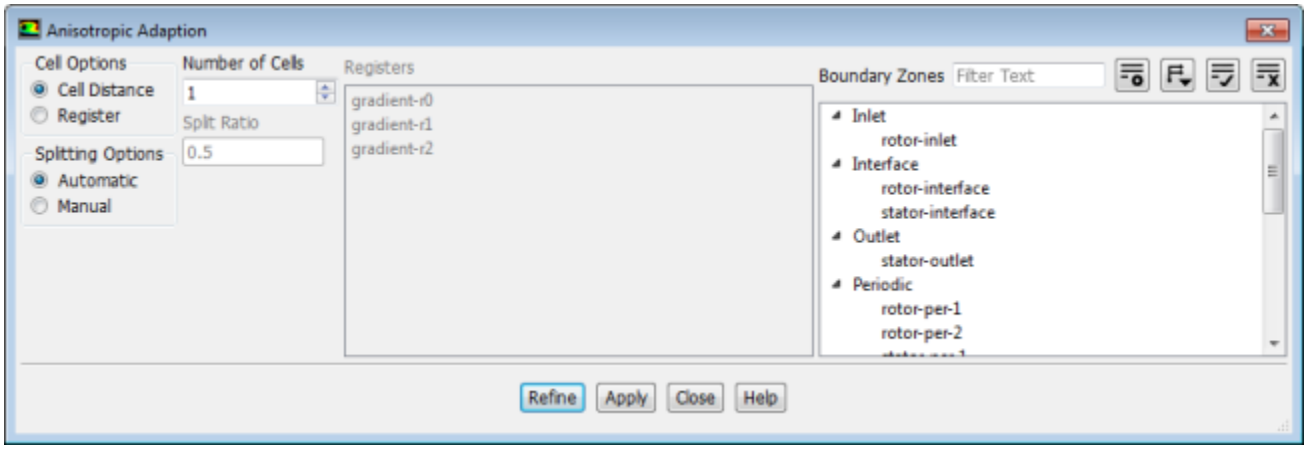
End Hold
specifies the number of video frames to hold the last animation frame. You may want to hold the last animation frame for about 5 seconds to provide closure.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.2.7. Anisotropic Adaption Dialog Box

The **Anisotropic Adaption** dialog box allows you to perform anisotropic adaption for certain cell types. See Anisotropic Adaption (p. 1697) for details.

Setting Up Domain → Adapt → Mark/Adapt Cells → Anisotropic...



Controls

Cell Options
controls the marking of boundary layer cells.

Cell Distance
indicates adaption using the distance of the marked cell to the boundary zone.

Register
indicates adaption using an existing register.

Splitting Options
control how the splitting ratio is computed.

Automatic
allows the ratio to be computed automatically from the mesh.

Manual
allows you to enter a specific value for the ratio in the **Split Ratio** field.

Number of Cells
indicates the number of cells to be adapted and marked (available only when the **Cell Distance** option is enabled).

Split Ratio
allows you to specify a value for the split ratio (available only when the **Manual** splitting option is enabled).

Registers
provides a list of existing registers (available only when the **Register** option is enabled).

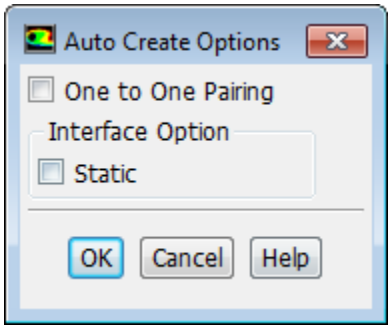
Boundary Zones
contains a list of available boundary zones.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Refine
refines the marked cells.

38.2.8. Auto Create Options Dialog Box

The **Auto Create Options** dialog box allows you to define settings that you want applied to the mesh interfaces that are created through the automatic method (for additional information, see Using a Non-Conformal Mesh in ANSYS Fluent (p. 164)). It is opened by clicking the **Options...** button in the **Unassigned Interface Zones** group box of the Mesh Interfaces Dialog Box (p. 2732).



Controls

One to One Pairing
specifies that the resulting mesh interfaces have a single zone assigned to each side. Using this option may decrease your computational expense and help with troubleshooting later on, though it may produce a larger number of mesh interfaces for you to manage.

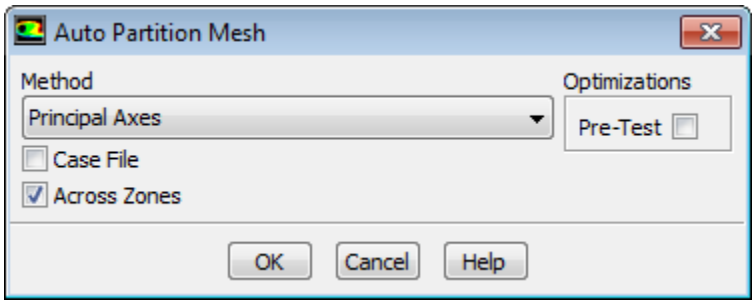
Interface Option
contains an option related to the interface type.

Static
reduces the memory usage and processing time for interface creation (as well as for solution), especially when you are creating interfaces using many unassigned interface zones. This option will only produce correct results if the interface zones do not move or deform relative to each other at the interfaces.

38.2.9. Auto Partition Mesh Dialog Box

The **Auto Partition Mesh** dialog box allows you to set the parameters for automatic partitioning when reading an unpartitioned mesh into the parallel solver. See Partitioning the Mesh Automatically (p. 2031) for details.

Parallel → General → Auto Partition...



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Method

contains a drop-down list of the recursive partition methods that can be used to create the mesh partitions. The choices include the **Cartesian Axes**, **Cartesian Strip**, **Cartesian X-Coordinate**, **Cartesian Y-Coordinate**, **Cartesian Z-Coordinate**, **Cartesian R Axes**, **Cartesian RX-Coordinate**, **Cartesian RY-Coordinate**, **Cartesian RZ-Coordinate**, **Cylindrical Axes**, **Cylindrical R-Coordinate**, **Cylindrical Theta-Coordinate**, **Cylindrical Z-Coordinate**, **Metis**, **Polar Axes**, **Polar R-Coordinate**, **Polar Theta-Coordinate**, **Principal Axes**, **Principal Strip**, **Principal X-Coordinate**, **Principal Y-Coordinate**, **Principal Z-Coordinate**, **Spherical Axes**, **Spherical Rho-Coordinate**, **Spherical Theta-Coordinate**, and **Spherical Phi-Coordinate** techniques, which are described in Mesh Partitioning Methods (p. 2046).

Case File

allows you to use a valid existing partition section in a case file (that is, one where the number of partitions in the case file divides evenly into the number of compute nodes). You need to turn off the **Case File** option only if you want to change other parameters in the **Auto Partition Mesh** dialog box.

Across Zones

allows partitions to cross zone boundaries (the default). If turned off, it will restrict partitioning to within each cell zone. This is recommended *only* when cells in different zones require significantly different amounts of computation during the solution phase, for example if the domain contains both solid and fluid zones.

Optimizations

contains an option to enable pre-testing.

Pre-Test

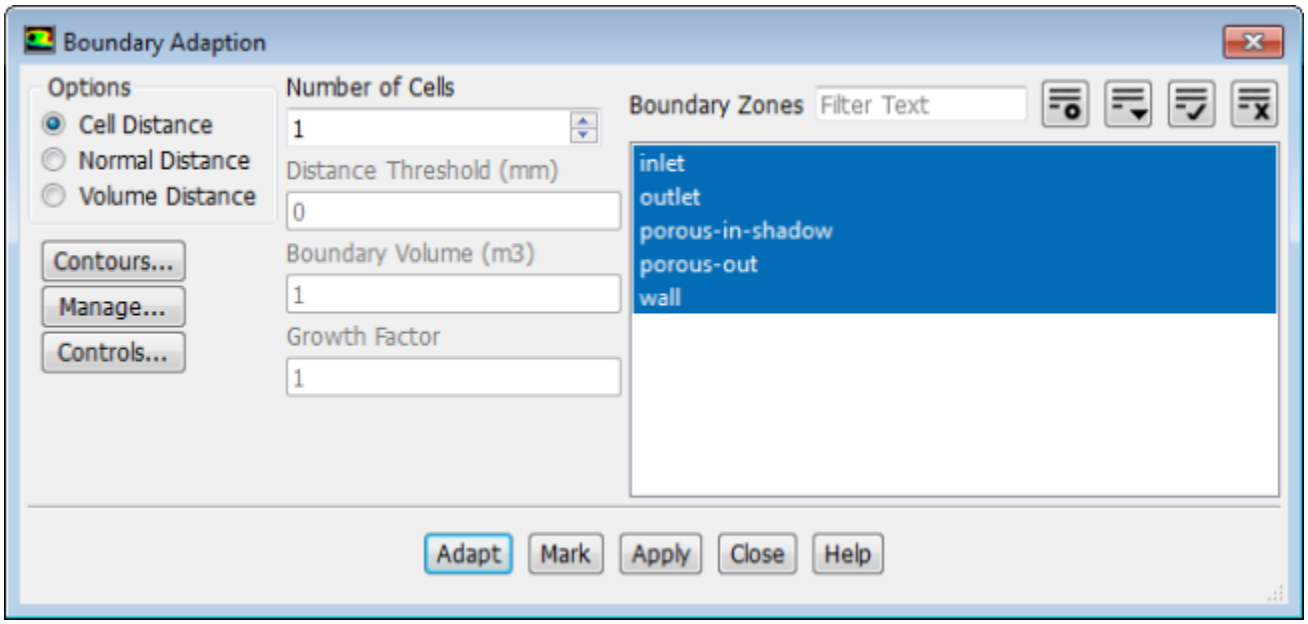
instructs Fluent to test all coordinate directions and choose the one which yields the fewest partition interfaces for the final bisection. Note that this option is available only when you choose **Principal Axes** or **Cartesian Axes** as the partitioning method.

38.2.10. Boundary Adaption Dialog Box

The **Boundary Adaption** dialog box allows you to mark or refine boundary cells on selected boundary zones. See Boundary Adaption (p. 1685) for details.

Setting Up Domain → Adapt → Mark/Adapt Cells → Boundary...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options
contains three different methods for boundary adaption:

Cell Distance
enables adaption based on a cell's distance from the boundary, measured in number of cells. See Boundary Adaption Based on Number of Cells (p. 1686) for details.

Normal Distance
enables adaption based on a cell's normal distance from the boundary. See Boundary Adaption Based on Normal Distance (p. 1686) for details.

Volume Distance
enables adaption based on a target boundary volume and growth factor. See Boundary Adaption Based on Target Boundary Volume (p. 1687) for details.

Contours...
opens the Contours Dialog Box (p. 2676), which you can use to determine the appropriate parameters for the boundary adaption.

Manage...
opens the Manage Adaption Registers Dialog Box (p. 2728), which allows you to display and manipulate adaption registers that are generated using the **Mark** command.

Controls...
opens the Mesh Adaption Controls Dialog Box (p. 2730), which allows you to control certain aspects of the adaption process.

Number of Cells
sets the maximum boundary cell distance for adaption (used with the **Cell Distance** option.

Distance Threshold
sets the maximum normal distance for adaption (used with the **Normal Distance** option).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Boundary Volume
sets the boundary volume $V_{boundary}$ in Equation 29.1 (p. 1687) (used with the **Volume Distance** option).

Growth Factor
sets the exponential growth factor α in Equation 29.1 (p. 1687) (used with the **Volume Distance** option).

Boundary Zones
contains a selectable list of zones on which you can refine. The boundary cells associated with the zones that you select will be refined.

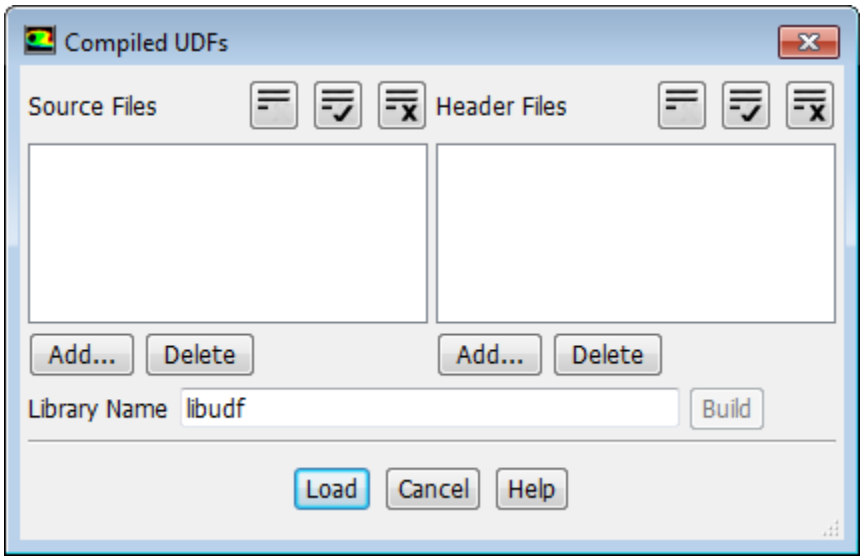
Adapt
refines the cells with edges/faces on the zones selected in the **Boundary Zones** list.

Mark
marks the boundary cells associated with the zones selected in the **Boundary Zones** list for refinement. This command produces an adaption register. For information on using adaption registers, see Manipulating Adaption Registers (p. 1701).

38.2.11. Compiled UDFs Dialog Box

The **Compiled UDFs** dialog box allows you to open a library of compiled user-defined functions. See the separate Fluent Customization Manual for details.

User Defined → User Defined → Functions → Compiled...



Controls

Source Files
contains a list of source files.

Header Files
contains a list of header files.

Add...
opens the **Select File** dialog box.

Delete
deletes the selected file from the list.

Library Name
specifies the name of the library to be created.

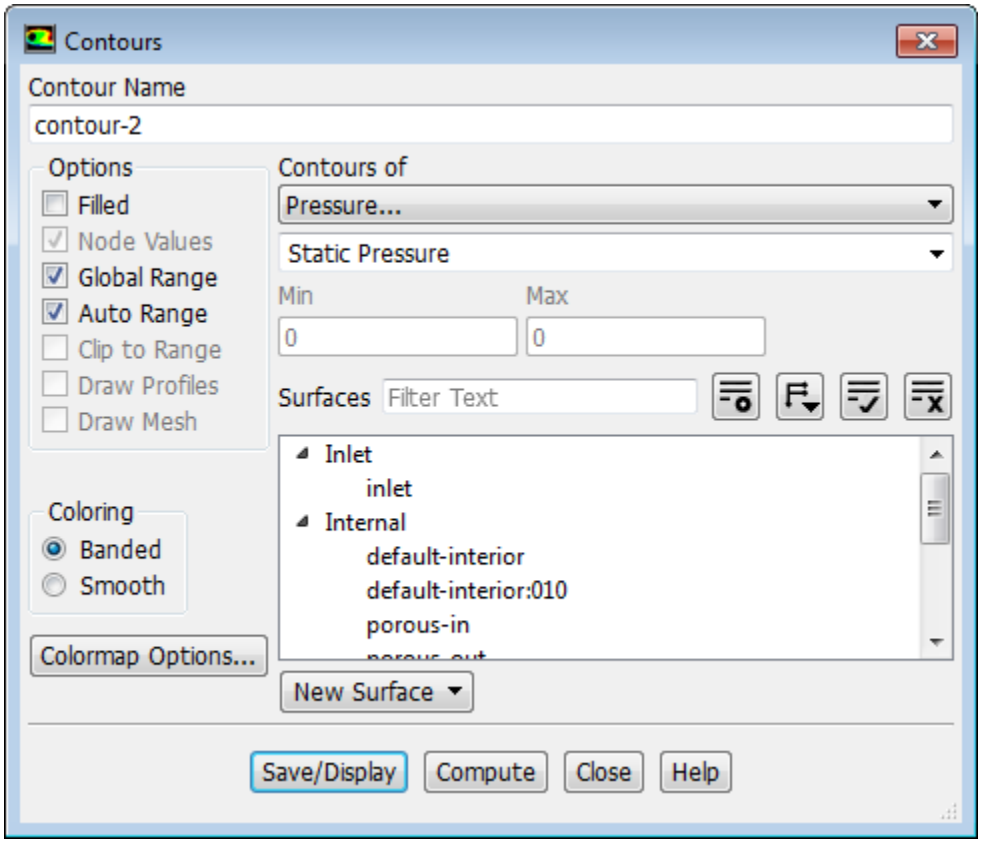
Build
builds the library and compiles the user-defined function.

Load
opens the specified library and loads the user-defined function.

38.2.12. Contours Dialog Box

The **Contours** dialog box controls the display of contour and profile plots. See Displaying Contours and Profiles (p. 1756) for details about the items below.

Postprocessing → Graphics → Contours → New...



Controls

Contour Name
is the name for a contour plot definition. You can specify a name or use the default name **`contour-id`**. This control appears only for contour plot definitions.

Options
contains the check buttons that set various contour display options.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Filled
toggles between filled contours and line contours.

Node Values
toggles between using scalar field values at nodes and at cell centers for computing the contours. When the **Filled** option is off, **Node Values** is always on. See Choosing Node or Cell Values (p. 1761) for details.

Global Range
toggles between basing the minimum and maximum values on the range of values on the selected surfaces (off), and basing them on the range of values in the entire domain (on, the default).

Auto Range
toggles between automatic and manual setting of the contour range. Any time you change the **Contours of** selection, **Auto Range** is reset to on.

Clip to Range
determines whether or not values outside the prescribed **Min**/ **Max** range are contoured when using **Filled** contours. If selected, values outside the range will not be contoured. If not selected, values below the **Min** value will be colored with the lowest color on the color scale, and values above the **Max** value will be colored with the highest color on the color scale. See Specifying the Range of Magnitudes Displayed (p. 1759) for details.

Draw Profiles
causes the addition of a profile plot to the contour plot. The Profile Options Dialog Box (p. 2556) is opened when **Draw Profiles** is selected.

Draw Mesh
toggles between displaying and not displaying the mesh. The Mesh Display Dialog Box (p. 2163) is opened when **Draw Mesh** is selected.

Levels
sets the number of contour levels that are displayed.

Setup
indicates the ID number of the contour setup. For frequently used combinations of contour fields and options, you can store the information needed to generate the contour plot by specifying a **Setup** number and setting up the desired information in the **Contours** dialog box. See Storing Contour Plot Settings (p. 1762) for details.

Contours of
contains a list from which you can select the scalar field to be contoured.

Min
shows the minimum value of the scalar field. If **Auto Range** is off, you can set the minimum by typing a new value.

Max
shows the maximum value of the scalar field. If **Auto Range** is off, you can set the maximum by typing a new value.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surfaces
contains a list from which you can select the surfaces on which to draw contours. For 2D cases, if no surface is selected, contouring is done on the entire domain. For 3D cases, you must always select at least one surface.

New Surface
is a drop-down list button that contains a list of surface options:

Point
opens the Point Surface Dialog Box (p. 2780).

Line/Rake
opens the Line/Rake Surface Dialog Box (p. 2727).

Plane
opens the Plane Surface Dialog Box (p. 2777).

Quadric
opens the Quadric Surface Dialog Box (p. 2781).

Iso-Surface
opens the Iso-Surface Dialog Box (p. 2721).

Iso-Clip
opens the Iso-Clip Dialog Box (p. 2720).

Display
draws the contours in the active graphics window.

Compute
calculates the scalar field and updates the **Min** and **Max** values (even when **Auto Range** is off).

Save/Display
plots the contour in the active graphics window and saves the contour plot definition. This button appears only for contour plot definitions and replaces the **Display** button.

38.2.13. Convergence Conditions Dialog Box

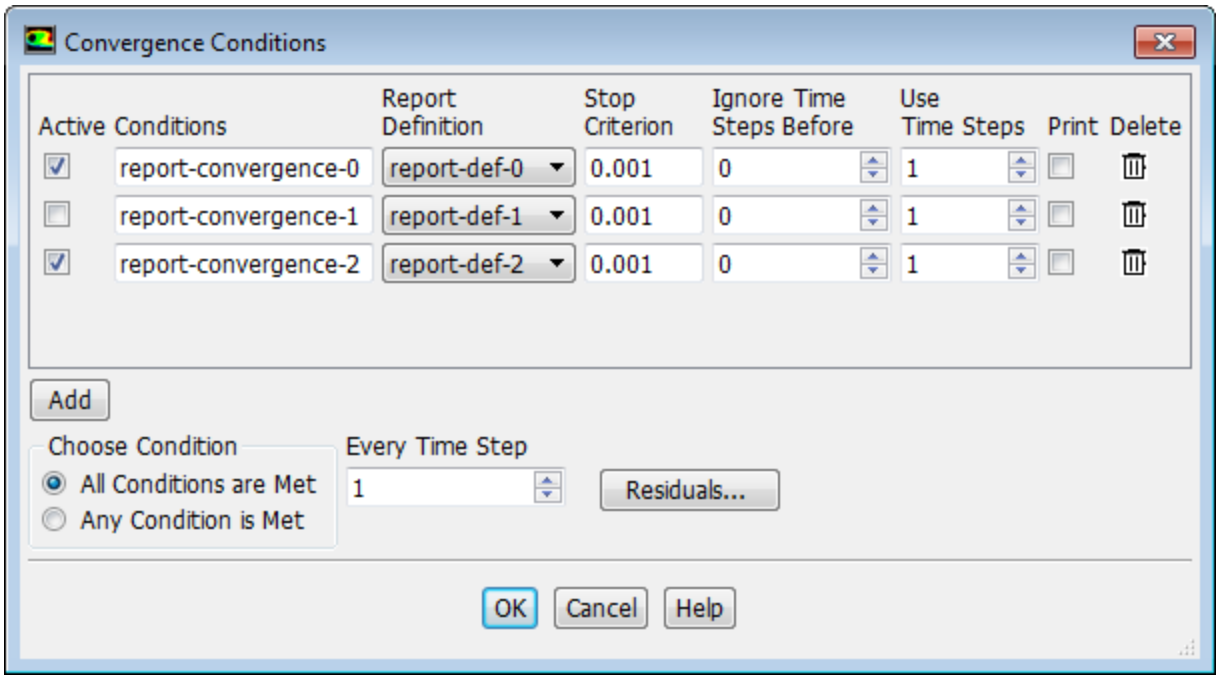
The convergence conditions facility allows you to set convergence conditions on the solution based on the values from report definitions (surface, volume, lift, drag, and so on). See Convergence Conditions (p. 1635) for details on setting up the **Convergence Conditions** dialog box.

Note

If you are solving a transient case, the **Convergence Conditions** dialog box will relabel some fields since the transient case uses time-steps rather than iterations. These alternate labels are indicated below.

Solving → Reports → Conditions...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Active
check box for deactivating/activating individual convergence conditions.

Conditions
name of the convergence condition.

Report Definition
name of the report definition used in judging convergence.

Stop Criterion
specifies the criterion below which the solution is considered to be converged.

Ignore Iterations Before | Ignore Time Steps Before
ignores the first few iterations/time-steps if you expect the solution to fluctuate initially.

Use Iterations | Use Time Steps
specifies the number of previous iterations/time-steps to be included in the report definition convergence check.

Choose Condition
to select the convergence conditions.

All Conditions are Met
The solution is considered to be converged if all of the convergence conditions' criteria are satisfied.

Any Condition is Met
The solution is considered to be converged if any of the convergence conditions' criteria is satisfied.

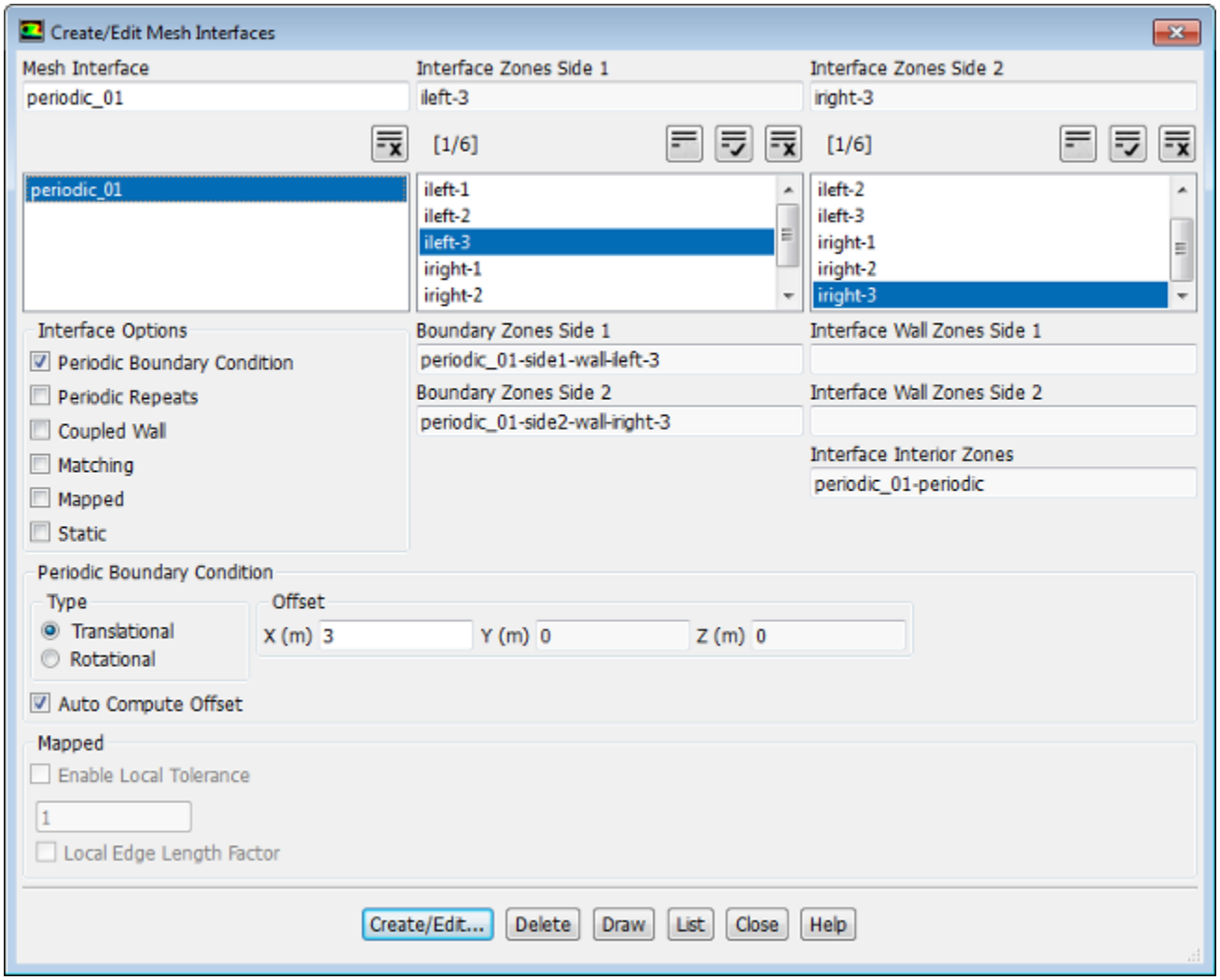
Every Iteration | Every Time-Step
to select how often convergence checks are done.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.2.14. Create/Edit Mesh Interfaces Dialog Box

The **Create/Edit Mesh Interfaces** dialog box allows you to manually create mesh interfaces for use with sliding meshes (see Using Sliding Meshes (p. 633)) or multiple reference frames (see Mesh Setup for a Multiple Moving Reference Frame (p. 616)), or for meshes with non-conformal boundaries (see Non-Conformal Meshes (p. 150)). This dialog box requires you to decide which interface zones make up both sides of each mesh interface. While it is possible to create every type of mesh interface using this dialog box, it is only necessary when you want your interface to use the periodic or periodic repeats option; for all other types it is more convenient to automatically create interfaces using the Mesh Interfaces Dialog Box (p. 2732), especially when you have many interface zones and/or are unfamiliar with their names / locations. For details, see Using a Non-Conformal Mesh in ANSYS Fluent (p. 164).

The **Create/Edit Mesh Interfaces** dialog box is opened by clicking the **Manual Create...** button in the Mesh Interfaces Dialog Box (p. 2732).



Controls

Mesh Interface

contains a text entry box in which you can set the name of the mesh interface, and a list from which you can select an existing mesh interface.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Interface Zones Side 1, Interface Zones Side 2
contain selectable lists for the interface zones that comprise the mesh interface and informational fields that show the names of the zones you selected in each list. (You cannot edit the top fields; the names in these fields will be the names of the zones you selected in the list below it.)

Interface Options
contains options related to the interface type.

Periodic Boundary Condition
allows you to create a non-conformal periodic boundary condition interface.

Periodic Repeats
should be enabled when each of the two cell zones has a single pair of conformal or non-conformal periodics adjacent to the interface. This option is typically used in conjunction with the sliding mesh model, when simulating the interface between a rotor and stator; it allows ANSYS Fluent to treat the interface between the sliding and non-sliding zones as periodic where the two zones do not overlap. For details, see The Periodic Repeats Option (p. 154).

Coupled Wall
allows you to specify that the interface acts as a thermally coupled wall.

Matching
is relevant if only interface internal zones should be created (that is, interior or wall / shadow pairs), since the interface zones on both sides are aligned. With the **Matching** option, even interface zones that are not perfectly aligned are treated as if they are; however, if the discrepancy between the interface zones on both sides exceeds default thresholds, then warning messages will be displayed. Note that the **Matching** option is compatible with the **Periodic Boundary Condition**, **Coupled Wall**, and **Static** options. For more information about the recommended uses of this option, see Matching Option (p. 157).

Mapped

enables an alternative approach for modeling coupled walls between zones. This approach is more robust (but less accurate) than the standard non-conformal interface formulations when the interface zones penetrate each other or have gaps between them.

Note

The **Mapped** option is not compatible with shell conduction.

Static

reduces the memory usage and processing time (for interface creation and solution), especially when there are many zones on both sides of the interface. This option will only produce correct results if the interface zones do not move or deform relative to each other at the interface, and it is not compatible with the **Periodic Boundary Condition**, **Periodic Repeats**, or **Mapped** options.

Boundary Zones Side 1, Boundary Zones Side 2
display the names of any wall boundary zones created by ANSYS Fluent during the process of creating the selected mesh interface. If the two interface zones overlap each other completely, then the wall boundaries are created but with zero faces.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Interface Wall Zones Side 1, Interface Wall Zones Side 2
display the names of any wall interface zones created by ANSYS Fluent during the process of creating the selected mesh interface.

Interface Interior Zones
displays the names of any interface interior zones created by ANSYS Fluent during the process of creating the selected mesh interface. This is used for embedded LES, when there is a need to be able to convert an interior zone into a RANS-LES interface.

Periodic Boundary Condition

Type
allows you to select a periodicity that is either **Translational** or **Rotational**.

Offset
is the offset coordinates or angle, depending on whether **Translational** or **Rotational** periodicity is selected. Note that when **Auto Compute Offset** is enabled, the **Offset** fields are not editable.

Auto Compute Offset
will result in ANSYS Fluent finding the offset. If this option is disabled, then you will have to provide the offset coordinates or angle in the required fields, depending on whether **Translational** or **Rotational** periodicity is selected.

Mapped

Enable Local Tolerance
(when enabled) allows you to overwrite any value specified in the **Mapped Tolerance** group box in the Interface Options Dialog Box (p. 2717), and is used on the selected interface only.

Local Edge Length Factor
(when enabled) allows you to specify the multiplier that Fluent uses with the smallest edge length of the interface zones to calculate the tolerance for mapping. Disabling **Local Edge Length Factor** allows you to specify an absolute value for the tolerance.

Create/Edit...
performs one of the following:

- If you have entered a new **Mesh Interface** name and selected unassigned zones from the **Interface Zones Side 1** and **Interface Zones Side 2** selection lists, a new mesh interface is created.
- If you have made a selection from the **Mesh Interface** list, the Edit Mesh Interfaces Dialog Box (p. 2696) will open so that you can revise the settings of existing mesh interfaces.

Delete
deletes the mesh interface selected under **Mesh Interface**.

Draw
allows you to display interface zones or mesh interfaces in the graphics window. Note that you can only select and display an interface zone from **Interface Zones Side 1** or **Interface Zones Side 2** if it is not yet assigned to an existing mesh interface. After a **Mesh Interface** is created, you can select the appropriate mesh interface and click the **Draw** button to display all of the zones under **Interface Zones Side 1** and **Interface Zones Side 2**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

List

prints information about the selected **Mesh Interface** in the console. When you click this button, ANSYS Fluent will list the two interface boundaries and (if you have initialized the solution) all new zones that were created (that is, wall and/or interior zones).

38.2.15. Custom Field Function Calculator Dialog Box

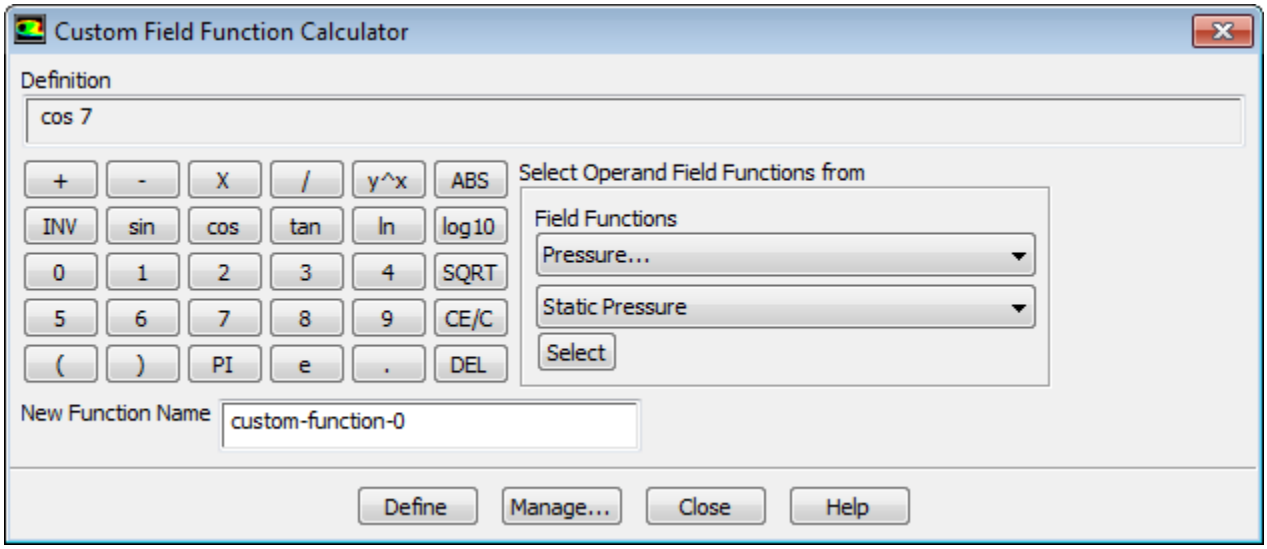
The **Custom Field Function Calculator** dialog box allows you to define custom field functions based on existing functions, using simple calculator operators. Any functions that you define will be added to the list of default flow variables and other field functions provided by ANSYS Fluent.

Important

Recall that you must enter all constants in the function definition in SI units.

See Creating a Custom Field Function (p. 2004) for details about the items below.

User Defined → Field Functions → Custom...



Controls

Definition

displays the function that you are currently defining. As you select each item from the **Field Functions** list or the calculator keypad, it will appear in the **Definition** text entry box. You *cannot* edit the contents of this box directly; if you want to delete part of a function, use the **Delete** button on the keypad.

(Calculator Buttons)

are buttons that perform calculator operations. When you select a calculator button (by clicking on it), the appropriate symbol will appear in the **Definition** text entry box.

Select Operand Field Functions from

contains the available field functions and the means for selecting them.

Field Functions

contains a list from which you can select a variable to be used in the definition of a new function.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select
enters the variable that is currently selected in the **Field Functions** list in the **Definition** field.

New Function Name
specifies the name of the function you are defining. Should you decide to change the function name after you have defined the function, you can do so in the Field Function Definitions Dialog Box (p. 2704), which you can open by clicking on the **Manage...** button.

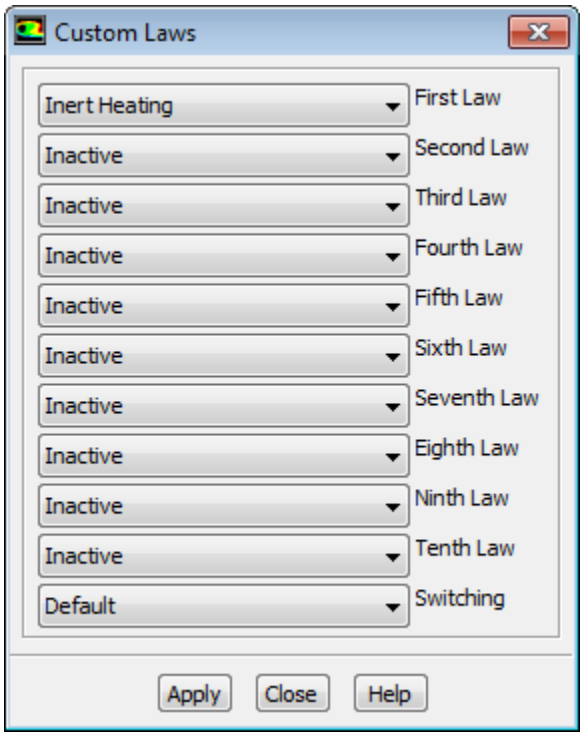
Define
creates the function and adds it to the list of **Custom Field Functions** within the drop-down list of available field functions. The **Define** button is grayed out after you create a new function or if the **Definition** field is empty.

Manage...
opens the Field Function Definitions Dialog Box (p. 2704), which enables you to check, rename, save, load, and delete custom field functions.

38.2.16. Custom Laws Dialog Box

The **Custom Laws** dialog box is used to incorporate user-defined functions (see the separate Fluent Customization Manual for details) in place of the default physical laws (1 through 6) used in the heat/mass transfer calculations.

The **Custom Laws** dialog box is opened from the Set Injection Properties Dialog Box (p. 2796).



Controls

First Law, Second Law, Third Law, Fourth Law, Fifth Law, Sixth Law
contain drop-down lists in which you can choose a user-defined particle law to replace the standard law.

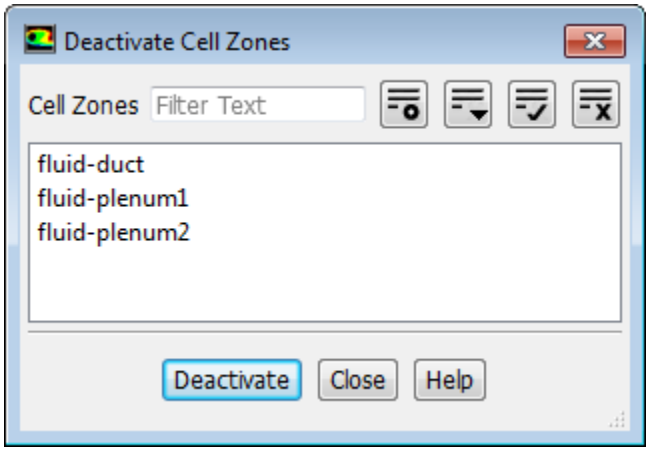
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Switching
contains a drop-down list in which you can select a user-defined function that customizes the way ANSYS Fluent switches between particle laws.

38.2.17. Deactivate Cell Zones Dialog Box

The **Deactivate Cell Zones** dialog box allows you to deactivate a single cell zone or multiple zones. See Deactivating Zones (p. 220) for details.

Setting Up Domain → Zones → Deactivate...



Controls

Cell Zones
contains a list of cell zones from which you can select the zone to be deactivated.

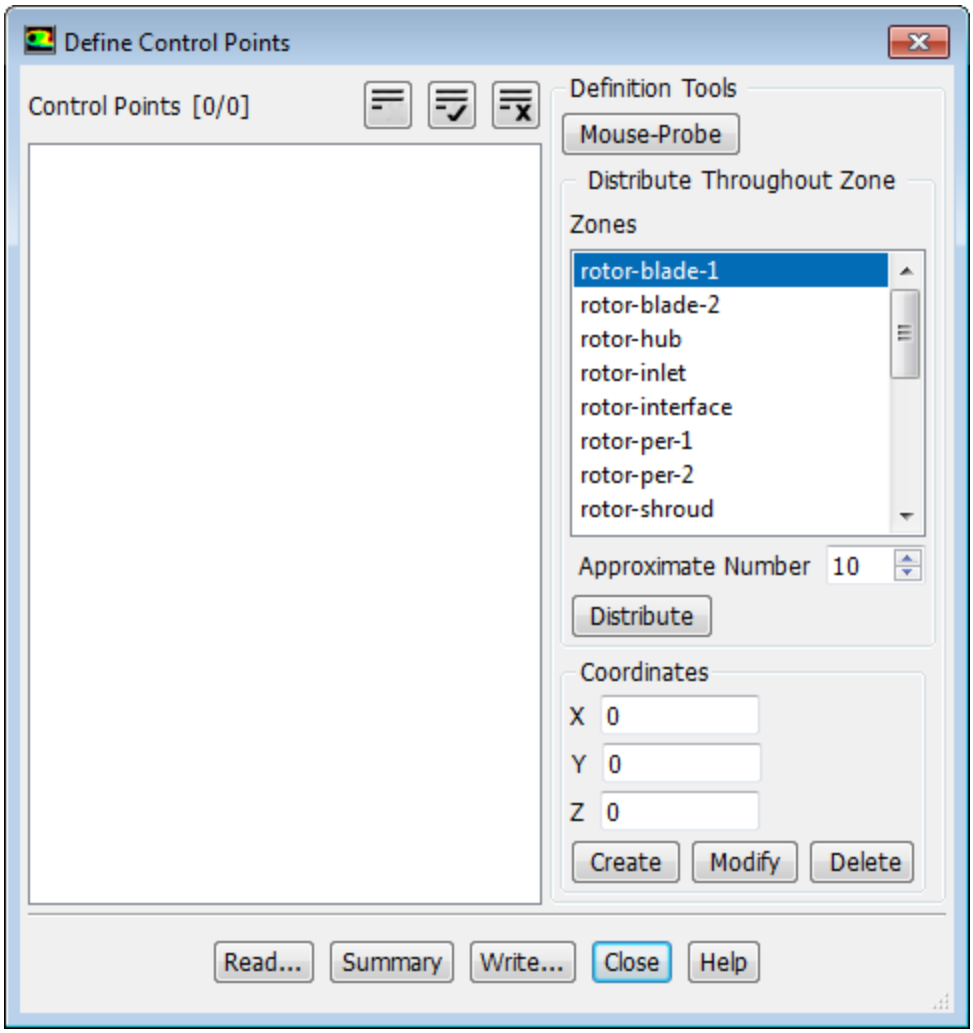
Deactivate
deactivates the selected cell zones.

38.2.18. Define Control Points Dialog Box

The **Define Control Points** dialog box allows you to create, modify, and delete control points for the mesh morpher/optimizer when the unstructured control point distribution is selected. See Using the Mesh Morpher/Optimizer (p. 2125) for details about using this dialog box.

The **Define Control Points** dialog box is opened from the Mesh Morpher/Optimizer Dialog Box (p. 2734).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Control Points
lists the control points you have created and allows you to modify and delete them.

Definition Tools
provides tools for defining the control points.

Mouse-Probe
allows you to create control points by probing (using the right mouse button, by default) in the graphics window.

Distribute Throughout Zone
allows you to quickly create a number of control points and distribute them on mesh nodes throughout a specific boundary zone.

Zones
allows you to select the boundary zone throughout which you want to distribute control points.

Approximate Number
allows you to specify the approximate number of control points you want to distribute throughout a zone. Note that due to the method by which the control points are distributed, the actual number may exceed the value you enter.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Distribute
creates a number of control points on mesh nodes throughout the specified zone (within the bounding box) with a distribution that is based on the distribution of the cell faces in that zone.

Coordinates
allows you to view and modify the coordinates of control points, as well as to create new control points and to delete existing control points.

X, Y, Z
displays the coordinates of the selected **Control Point**, and allows you to define new ones and edit existing ones.

Create
creates a new control point at the currently displayed **X**, **Y**, and **Z** coordinates.

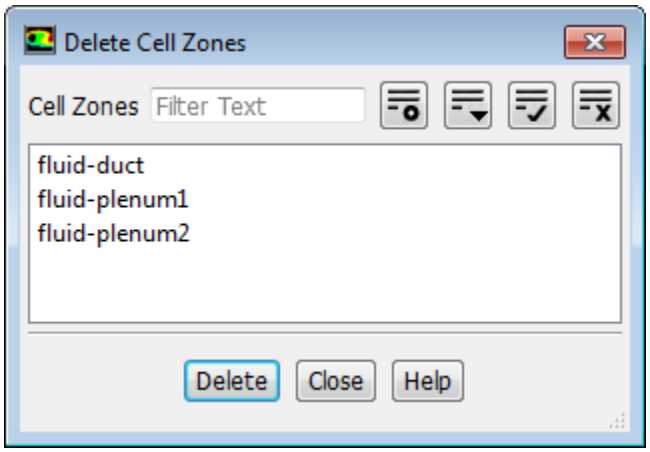
Modify
revises the **X**, **Y**, and **Z** coordinates of the selected **Control Point**.

Delete
deletes the selected **Control Point**.

38.2.19. Delete Cell Zones Dialog Box

The **Delete Cell Zones** dialog box allows you to delete a single cell zone or multiple zones. See Deleting Zones (p. 219) for details.

Setting Up Domain → Zones → Delete...



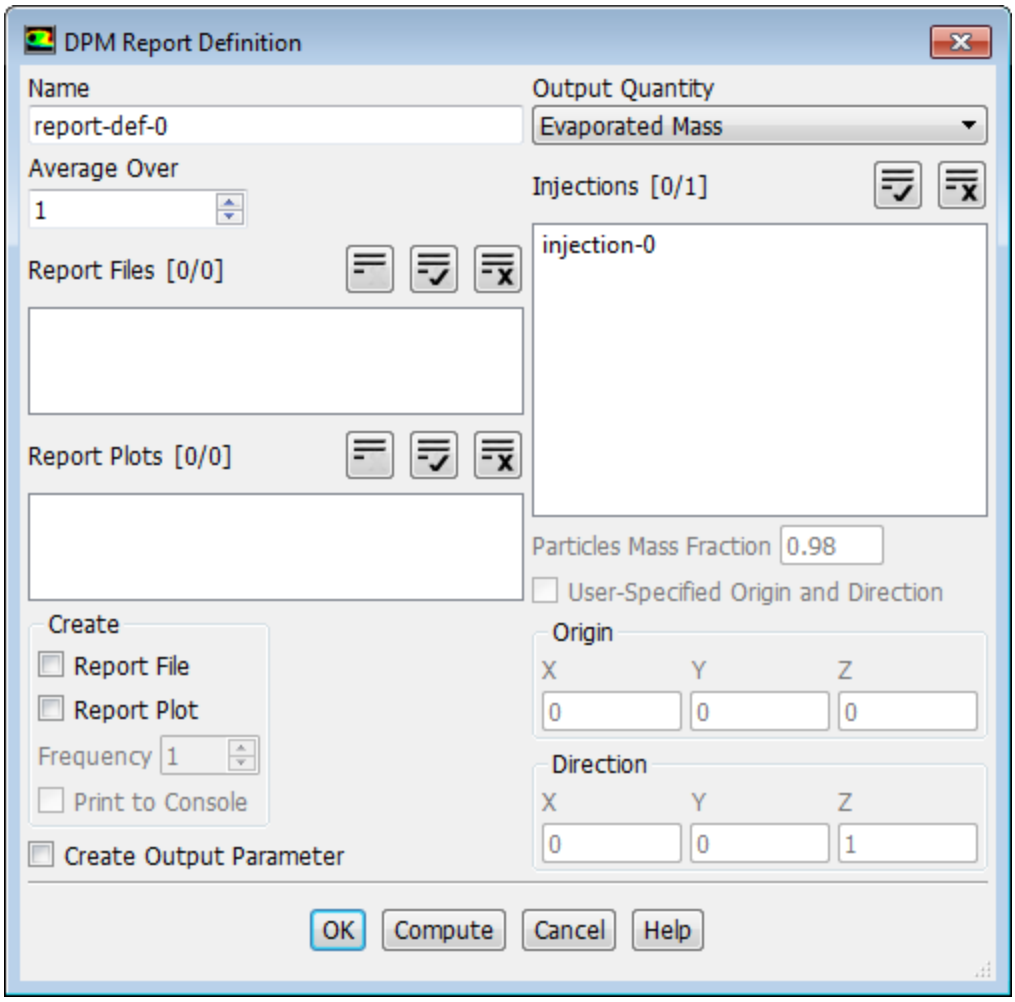
Controls

Cell Zones
contains a list of cell zones from which you can select the zone to be deleted.

Delete
deletes the selected cell zones.

38.2.20. DPM Report Definition Dialog Box

Solving → Reports → Definitions → New → DPM Report



Controls

Average Over

(optional) To have Fluent calculate a running average for the **DPM Report Definition** you can enter a positive integer greater than 1 (the default) for **Average Over**.

Specifying a number greater than 1 means that Fluent will print, plot, and write the running average value of the selected variable instead of the current value of the same variable.

The value reported is averaged over the last N *iterations/time steps, where* N *is your specified* ***Average Over*** *value. When the iteration/time step number is lower than* N*, Fluent calculates the average of the available variable values.*

Report Files

lists all of the report files where you can write the report definition data.

Report Plots

lists all of the report plots where you can plot the report definition data (assuming this report has the same units as the other report definitions included in the selected report plot).

Create

Report File

creates a new report file that includes this report definition.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Report Plot
creates a new report plot that includes this report definition.

Frequency
specifies how frequently the report definition is written, plotted, or printed to the console.

Print to Console
prints the value of the report definition to the console.

Create Output Parameter
creates an output parameter for this report definition with the name `<report definition name>-op`.

Output Quantity

Injected Mass
(unsteady particle tracking only) total mass injected into the domain.

Mass in Domain
(unsteady particle tracking only) total mass present in the domain.

Escaped Mass
reports the mass that has left the domain through a certain boundary or boundaries.

Evaporated Mass
reports the total evaporated mass for the selected injection(s). The quantity accounts for fully evaporated particles as well as all mass that has evaporated from any other particles. For unsteady tracking, this includes particles that are still present in the domain.

Penetration Length
(unsteady particle tracking only) reports the total penetration length in meters.

Injections
lists the injections that you can select for discrete phase reporting. For the **Escaped Mass** output quantity, this item is available only if you have enabled the `report/dpm-zone-summaries-per-injection` text command. Note that for unsteady particle tracking, if you want to report the mass of escaped particles per injection, this text command must be enabled before any particles are injected into the domain.

Boundaries
contains a selectable list of valid boundary zones for escaped mass reporting.

Show Mass Flow/Change Rate
(unsteady particle tracking only) if enabled, specifies that the mass flow rate will be reported (default). Otherwise the total mass will be reported. This item is not available for the **Penetration Length** report.

Particles Mass Fraction
the mass fraction expected between the injection nozzle and the penetration length. This item is available only for the **Penetration Length** reports.

User-Specified Origin and Direction
once enabled, it allows you to specify the **Origin** and **Direction** of the selected injection(s). This item is available only for the **Penetration Length** reports.

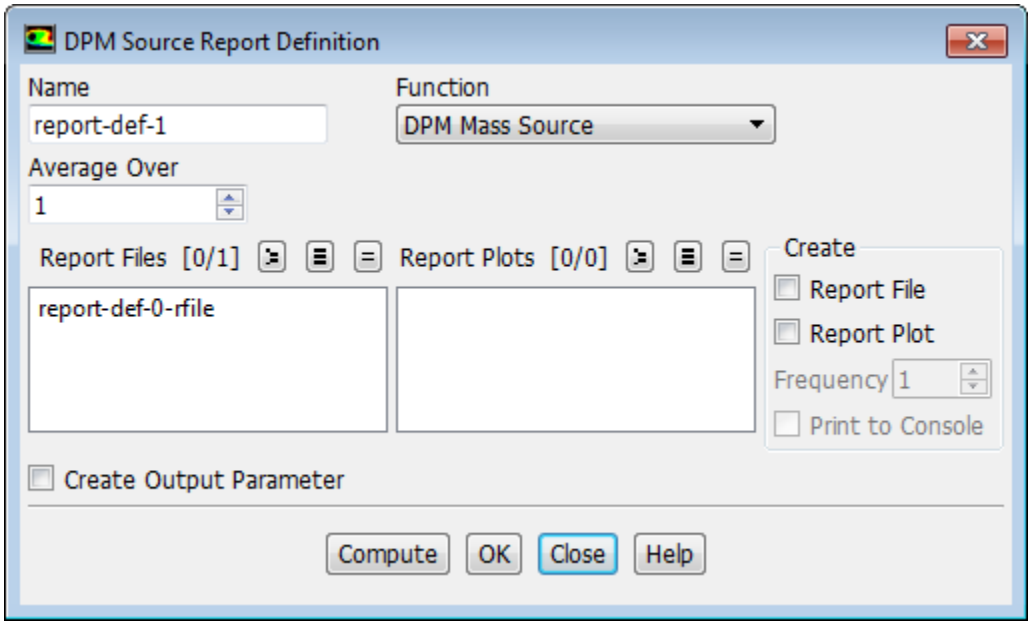
OK
creates the report definition.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Compute
computes the value of the report definition and prints to the console.

38.2.21. DPM Source Report Definition

Solving → Reports → Definitions → New → Flux Report → DPM Mass Source | DPM Enthalpy Source | DPM Sensible Enthapy Source



Controls

Function

DPM Mass Source
reports the mass contributions from the particle injections.

DPM Enthalpy Source
reports the heat contributions from the particle injections.

DPM Sensible Enthalpy Source
reports the sensible heat contributions from the particle injections.

Average Over
(optional) To have Fluent calculate a running average for the **DPM Source Report Definition** you can enter a positive integer greater than 1 (the default) for **Average Over**.

Specifying a number greater than 1 means that Fluent will print, plot, and write the running average value of the selected variable instead of the current value of the same variable.

The value reported is averaged over the last N *iterations/time steps, where* N *is your specified* ***Average Over*** *value. When the iteration/time step number is lower than* N*, Fluent calculates the average of the available variable values.*

Report Files
lists all of the report files where you can write the report definition data.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Report Plots
lists all of the report plots where you can plot the report definition data (assuming this report has the same units as the other report definitions included in the selected report plot).

Create Output Parameter
creates an output parameter for this report definition with the name `<report definition name>-op`.

Create

Report File
creates a new report file that includes this report definition.

Report Plot
creates a new report plot that includes this report definition.

Frequency
specifies how frequently the report definition is written, plotted, or printed to the console.

Print to Console
prints the value of the report definition to the console.

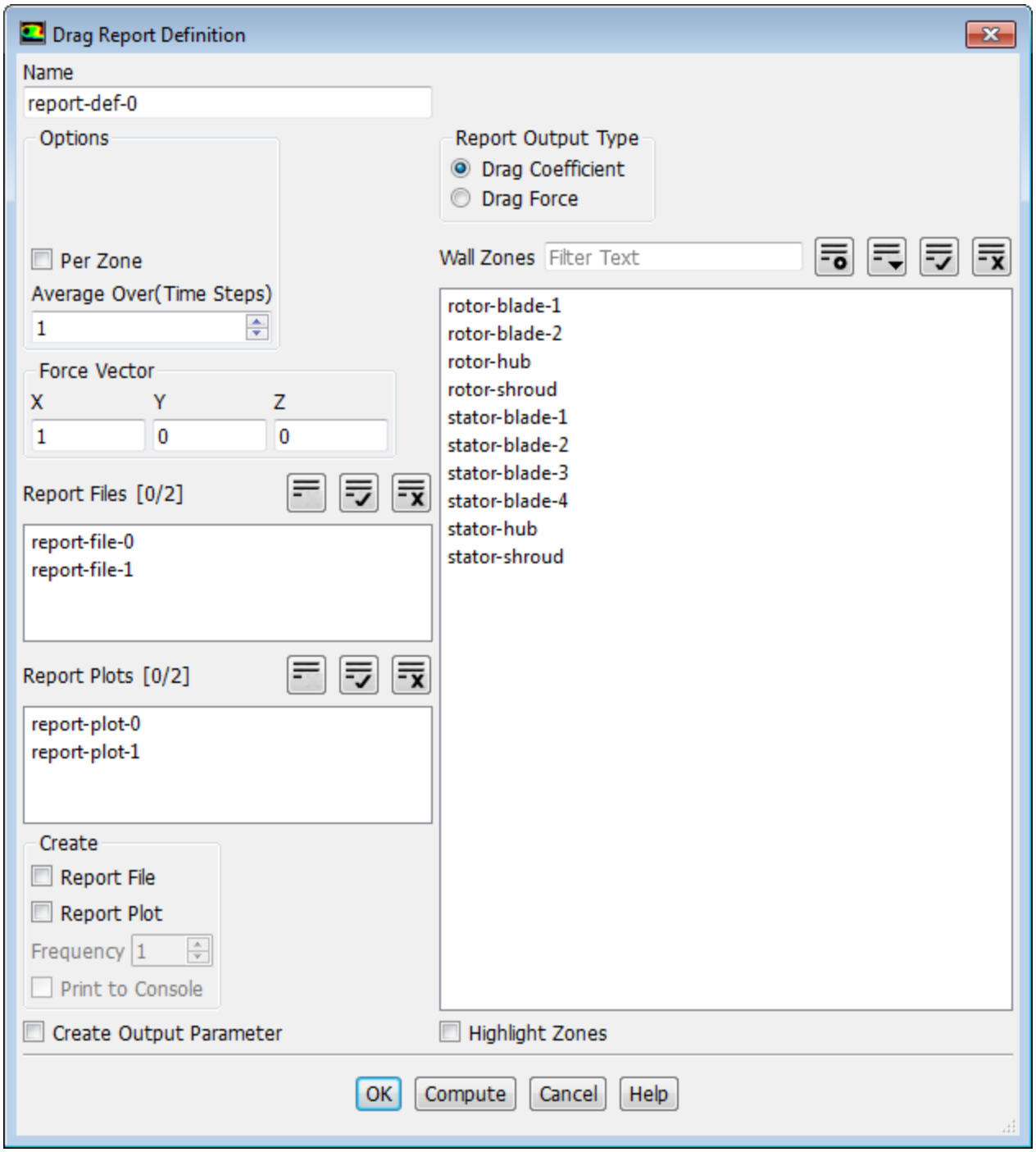
Compute
computes the value of the report definition and prints to the console.

OK
creates the report definition.

38.2.22. Drag Report Definition Dialog Box

Solving → Reports → Definitions → New → Force Report → Drag...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options

Per Zone
specifies whether or not the drag for multiple walls zones should be monitored on each wall zone separately.When this option is turned off (the default), Fluent computes and monitors the total drag for all of the selected walls zones combined together.

Average Over (Time-Steps), Average Over (Iterations)
(optional) To have Fluent calculate a running average for the **Drag Report Definition** you can enter a positive integer greater than 1 (the default) for **Average Over**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specifying a number greater than 1 means that Fluent will print, plot, and write the running average value of the selected variable instead of the current value of the same variable.

The value reported is averaged over the last N *iterations/time steps, where* N *is your specified* ***Average Over*** *value. When the iteration/time step number is lower than* N*, Fluent calculates the average of the available variable values.*

Force Vector
contains the components of the force vector.

X,Y,Z
are the components of the force vector along which the forces will be computed.

Report Files
lists all of the report files where you can write the report definition data.

Report Plots
lists all of the report plots where you can plot the report definition data (assuming this report has the same units as the other report definitions included in the selected report plot).

Create

Report File
creates a new report file that includes this report definition.

Report Plot
creates a new report plot that includes this report definition.

Frequency
specifies how frequently the report definition is written, plotted, or printed to the console.

Print to Console
prints the value of the report definition to the console.

Create Output Parameter
creates an output parameter for this report definition with the name `<report definition name>-op`.

Report Output Type

Drag Coefficient
specifies the report definition as a unitless quantity.

Drag Force
specifies that the report definition has units of force.

Wall Zones
lists the zones you can select for drag coefficient or drag force computation.

OK
creates the report definition.

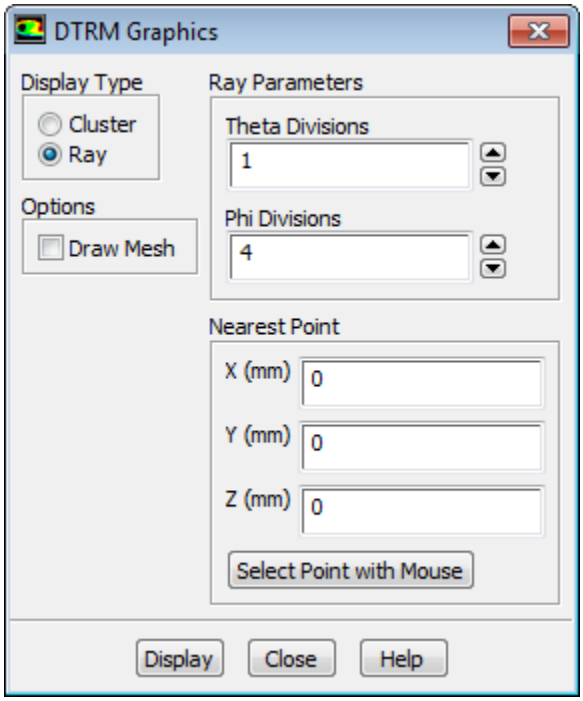
Compute
computes the value of the report definition at the selected zone(s) and prints to the console.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.2.23. DTRM Graphics Dialog Box

The **DTRM Graphics** dialog box allows you to display rays and clusters used by the DTRM. See Displaying Rays and Clusters for the DTRM (p. 883) for details.

Postprocessing → Model Specific → DTRM Graphics...



Controls

Display Type
contains options for the different items you can display.

Cluster
specifies the display of clusters.

Ray
specifies the display of rays.

Options
contains check buttons that control display options.

Draw Mesh
toggles between displaying and not displaying the mesh. The Mesh Display Dialog Box (p. 2163) is opened when **Draw Mesh** is selected.

Cluster Type
specifies whether **Surface** clusters or **Volume** clusters are to be displayed. This section of the dialog box appears when **Cluster** is selected as the **Display Type**.

Cluster Selection
contains options for cluster displays. This section of the dialog box appears when **Cluster** is selected as the **Display Type**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Display All Clusters
enables the display of all surface or volume clusters in the domain.

Ray Parameters
contains controls for displaying rays. This section of the dialog box appears when **Ray** is selected as the **Display Type**.

Theta Divisions, Phi Divisions
control the number of rays being traced. (See Controlling the Rays (p. 850).)

Nearest Point
specifies a point (**X,Y,Z**) near the cluster to be displayed (or the cluster from which the rays should start).

The **Nearest Point** controls are not available when **Display All Clusters** is selected under **Cluster Selection**.

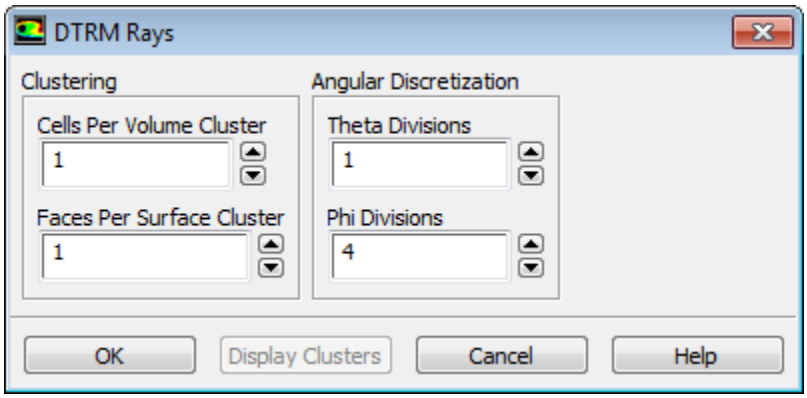
Select Point With Mouse
is an alternative method for specifying the **Nearest Point** using your mouse, by clicking the button.

Display
displays the specified cluster(s) or rays.

38.2.24. DTRM Rays Dialog Box

The **DTRM Rays** dialog box allows you to define the rays used by the discrete transfer radiation model (DTRM). It opens automatically when you click **OK** after selecting the **Discrete Transfer** model in the Radiation Model Dialog Box (p. 2190). See Setting Up the DTRM (p. 848) for details about the items below.

Setting Up Physics → Model Specific → DTRM Rays...



Controls

Clustering
contains parameters for the clusters (see Clustering in the Theory Guide).

Cells Per Volume Cluster, Faces Per Surface Cluster
control the number of radiating surfaces and absorbing cells. (See the explanation in Controlling the Clusters (p. 849).)

Angular Discretization
contains parameters for the ray traces (see Ray Tracing in the Theory Guide).

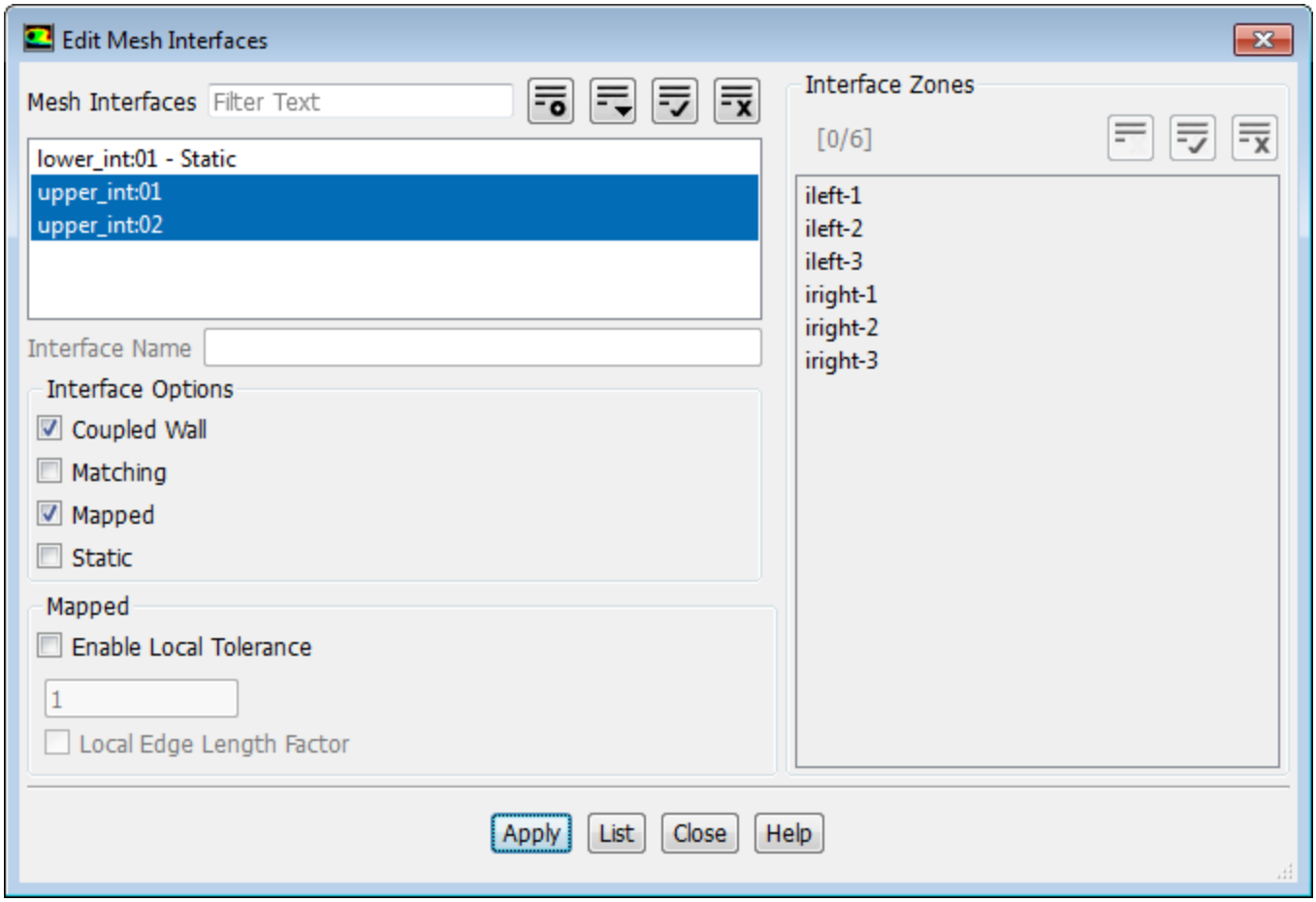
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Theta Divisions, Phi Divisions
control the number of rays being traced. (Guidelines are provided in Controlling the Rays (p. 850).)

Display Clusters
generates a graphical display of the clusters in the domain. (This item is available only after you have created or read a ray file.)

38.2.25. Edit Mesh Interfaces Dialog Box

The **Edit Mesh Interfaces** dialog box allows you to edit existing mesh interfaces (see Using a Non-Conformal Mesh in ANSYS Fluent (p. 164)). It is opened by making a selection from the **Mesh Interfaces** list in the Mesh Interfaces Dialog Box (p. 2732) and clicking **Edit...**.



Controls

Mesh Interfaces

allows you to select the mesh interfaces you would like to edit. Note that you can click the button to group the interfaces in the list by their currently enabled interface option(s), which allows you to select / deselect all those of a particular type by simply clicking the top-level branch.

Interface Name
edits the name of the mesh interface. This is only available when a single interface is selected.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Interface Options

contains options related to the interface type. These options can be applied when a single or multiple interfaces are selected.

Periodic Boundary Condition

allows you to create a non-conformal periodic boundary condition interface.

Periodic Repeats

should be enabled when each of the two cell zones has a single pair of conformal or non-conformal periodics adjacent to the interface. This option is typically used in conjunction with the sliding mesh model, when simulating the interface between a rotor and stator; it allows ANSYS Fluent to treat the interface between the sliding and non-sliding zones as periodic where the two zones do not overlap. For details, see The Periodic Repeats Option (p. 154).

Coupled Wall

allows you to specify that the interface acts as a thermally coupled wall.

Matching

is relevant if only interface internal zones should be created (that is, interior or wall / shadow pairs), since the interface zones on both sides are aligned. With the **Matching** option, even interface zones that are not perfectly aligned are treated as if they are; however, if the discrepancy between the interface zones on both sides exceeds default thresholds, then warning messages will be displayed. Note that the **Matching** option is compatible with the **Periodic Boundary Condition**, **Coupled Wall**, and **Static** options. For more information about the recommended uses of this option, see Matching Option (p. 157).

Mapped

enables an alternative approach for modeling coupled walls between zones. This approach is more robust (but less accurate) than the standard non-conformal interface formulations when the interface zones penetrate each other or have gaps between them.

Note

The **Mapped** option is not compatible with shell conduction.

Static

reduces the memory usage and processing time (for interface creation and solution), especially when there are many zones on both sides of the interface. This option will only produce correct results if the interface zones do not move or deform relative to each other at the interface, and it is not compatible with the **Periodic Boundary Condition**, **Periodic Repeats**, or **Mapped** options.

Periodic Boundary Conditions

Type

allows you to select a periodicity that is either **Translational** or **Rotational**.

Offset

is the offset coordinates or angle, depending on whether **Translational** or **Rotational** periodicity is selected. Note that when **Auto Compute Offset** is enabled, the **Offset** fields are not editable.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Auto Compute Offset
will result in ANSYS Fluent finding the offset. If this option is disabled, then you will have to provide the offset coordinates or angle in the required fields, depending on whether **Translational** or **Rotational** periodicity is selected.

Mapped

Enable Local Tolerance
(when enabled) allows you to overwrite any value specified in the **Mapped Tolerance** group box in the Interface Options Dialog Box (p. 2717), and is applied to the selected interface only.

Local Edge Length Factor
(when enabled) allows you to specify the multiplier that Fluent uses with the smallest edge length of the interface zones to calculate the tolerance for mapping. Disabling **Local Edge Length Factor** allows you to specify an absolute value for the tolerance.

Interface Zones
highlights the list of interface zones that comprise the selected mesh interface and allows you to edit the list (provided that the zones you select are unassigned and located appropriately). This functionality is only available when a single interface is selected.

List
prints information in the console about the first mesh interface you selected. When you click this button, ANSYS Fluent will list the two interface boundaries and (if you have initialized the solution) all new zones that were created (that is, wall and/or interior zones).

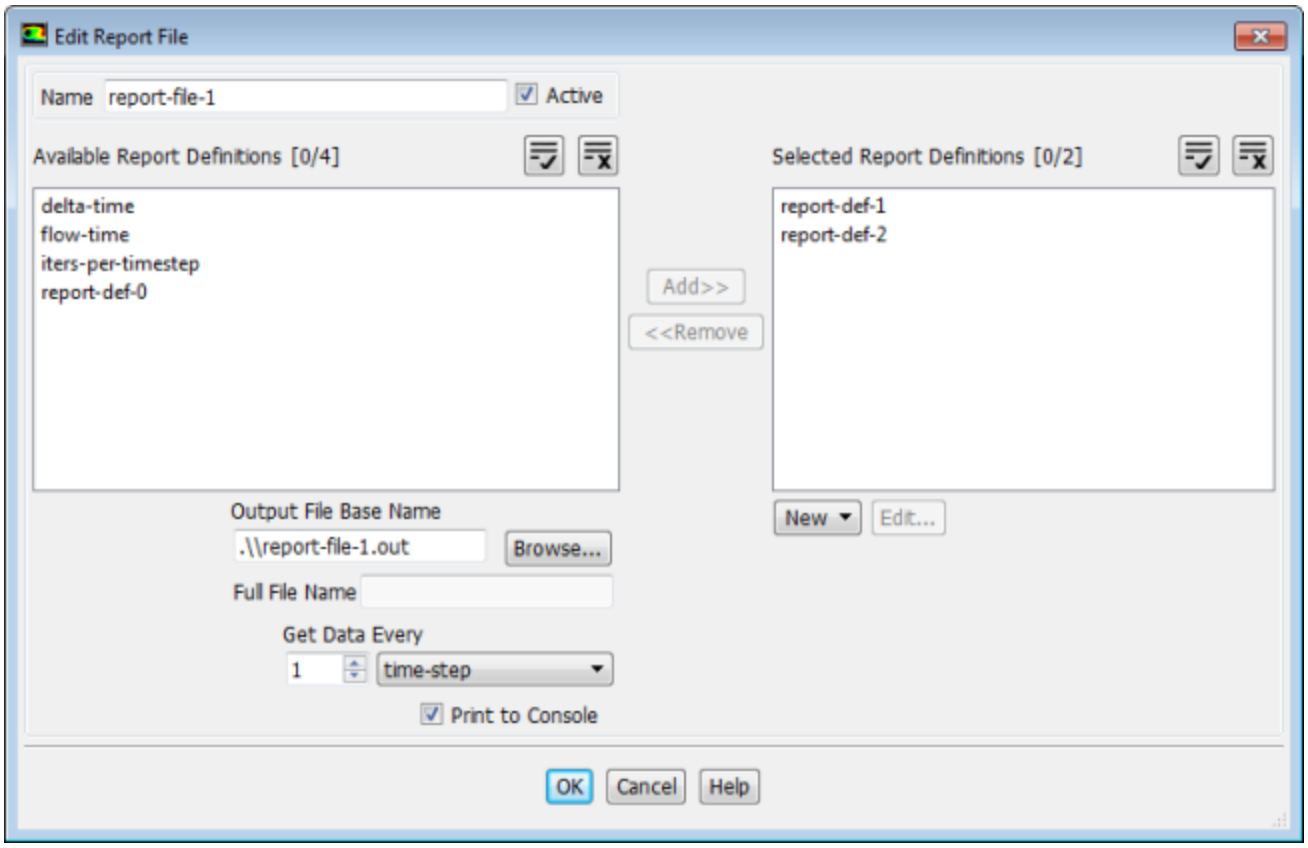
Apply
saves the revised settings for the selected mesh interfaces.

38.2.26. Edit Report File Dialog Box

Solving → **Reports** → **File...**

Select a report file in the **Report File Definition** dialog box and click **Edit...** to open the **Edit Report File** dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name
editable field displaying the name of the report file.

Active
check box (enabled by default) that determines if the file is written during the solution calculation.

Available Report Definitions
lists all of the existing report definitions not currently part of this report file.

Add>>
moves the selected report definition(s) to the **Selected Report Definitions** list.

<<Remove
moves the selected report definition(s) to the **Available Report Definitions** list.

Output File Base Name
allows you to specify the name used when the file is saved.

Browse...
opens the **Select File** dialog box, allowing you to specify the location where the report file will be saved.

Full File Name
displays the name and location of the report file once it has been saved.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Get Data Every
indicates the frequency that the report definition data is written to the file. The default value of 1 allows you to write to the file at every iteration or time-step. Time-step is only a valid choice you are calculating unsteady flow.

Print to Console
indicates whether or not the contents of the report file are printed to the console at the frequency specified in the **Get Data Every** field.

Selected Report Definitions
lists all of the existing report definitions currently included in this report file.

New
drop-down list allowing you to create new report definitions (surface, volume, force, flux, dpm, and user-defined).

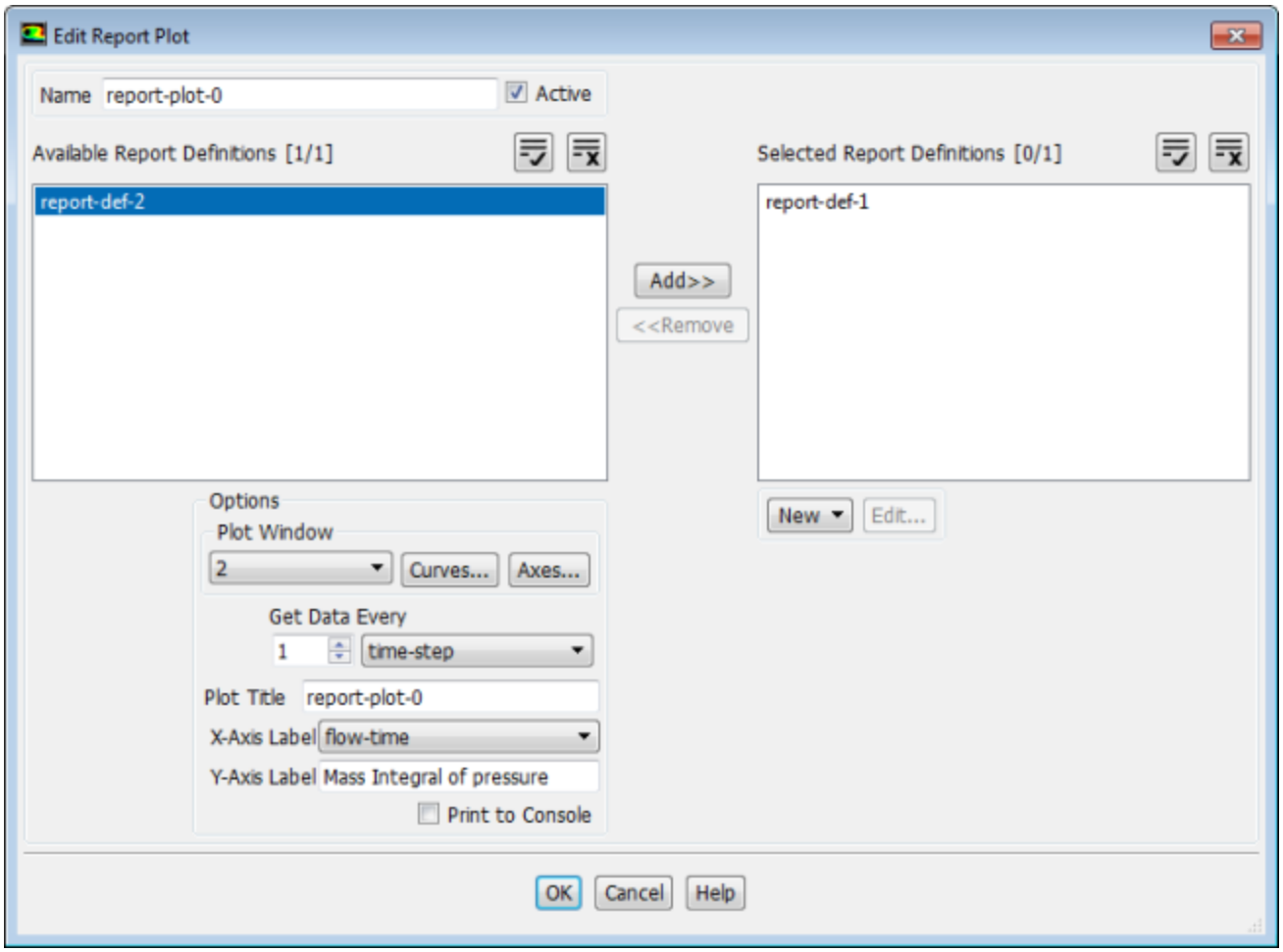
Edit...
allows you to modify the highlighted report definition in the **Selected Report Definition** list.

38.2.27. Edit Report Plot Dialog Box

Solving → **Reports** → **Plot...**

Select a report plot in the **Report Plot Definition** dialog box and click **Edit...** to open the **Edit Report Plot** dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name
editable field displaying the name of the report plot.

Active
check box (enabled by default) that determines if the plot is plotted during the solution calculation.

Available Report Definitions
lists all of the existing report definitions not currently part of this report plot.

Add>>
moves the selected report definition(s) to the **Selected Report Definitions** list.

<<Remove
moves the selected report definition(s) to the **Available Report Definitions** list.

Options

Plot Window
allows you to specify the graphics window for this report plot.

Curves...
opens the Curves Dialog Box (p. 2610).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Axes...
opens the Axes Dialog Box (p. 2608).

Get Data Every
indicates the frequency that the report definition data is plotted in the graphics window. The default value of 1 allows you to plot the data at every iteration or time-step. Time-step is only a valid choice you are calculating unsteady flow.

Plot Title
provide a title for the plot.

X-Axis Label
choose whether the X-axis is flow-time or time-step.

Y-Axis Label
enter a label for the Y-axis.

Print to Console
indicates whether or not the contents of the report plot are printed to the console at the frequency specified in the **Get Data Every** field.

Selected Report Definitions
lists all of the existing report definitions currently included in this report plot.

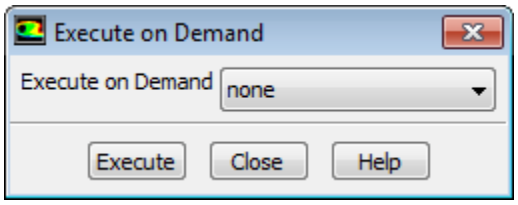
New
drop-down list allowing you to create new report definitions (surface, volume, force, flux, dpm, and user-defined).

Edit...
allows you to modify the highlighted report definition in the **Selected Report Definition** list.

38.2.28. Execute on Demand Dialog Box

The **Execute on Demand** dialog box allows you to run a specified user-defined function immediately. See the separate Fluent Customization Manual for details.

User Defined → User Defined → Execute on Demand...



Controls

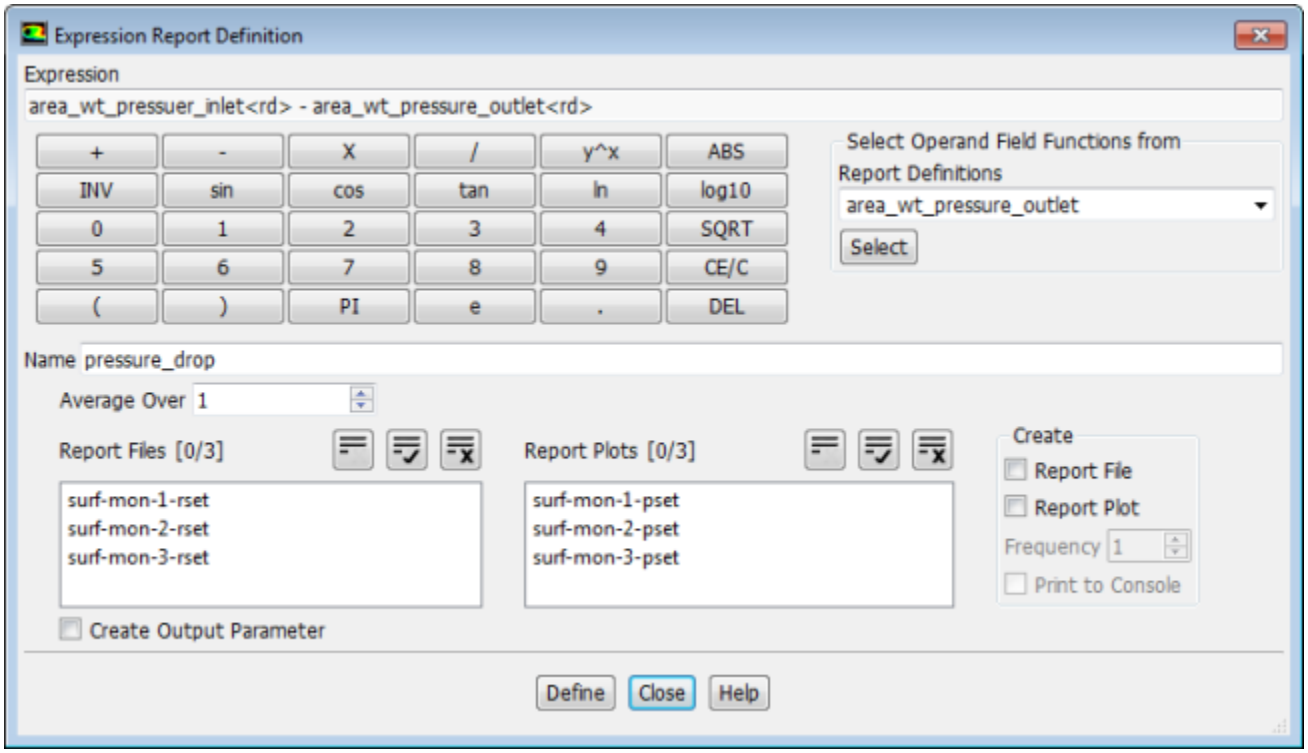
Execute on Demand
selects the user-defined function to be run.

Execute
runs the selected function.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.2.29. Expression Report Definition Dialog Box

Solving → Reports → Definitions → New → Expression



Controls

Expression
displays the formula of the expression as you create it.

Select Operand Field Functions from

Report Definitions
drop-down list containing all of the currently-defined report definitions.

Select
adds the listed report definition to the **Expression** formula.

Average Over
(optional) To have Fluent calculate a running average for the **Expression Report Definition** you can enter a positive integer greater than 1 (the default) for **Average Over**.

Specifying a number greater than 1 means that Fluent will print, plot, and write the running average value of the selected variable instead of the current value of the same variable.

The value reported is averaged over the last N *iterations/time steps, where* N *is your specified* ***Average Over*** *value. When the iteration/time step number is lower than* N*, Fluent calculates the average of the available variable values.*

Report Files
lists all of the report files where you can write the report definition data.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Report Plots
lists all of the report plots where you can plot the report definition data (assuming this report has the same units as the other report definitions included in the selected report plot).

Create

Report File
creates a new report file that includes this report definition.

Report Plot
creates a new report plot that includes this report definition.

Frequency
specifies how frequently the report definition is written, plotted, or printed to the console.

Print to Console
prints the value of the report definition to the console.

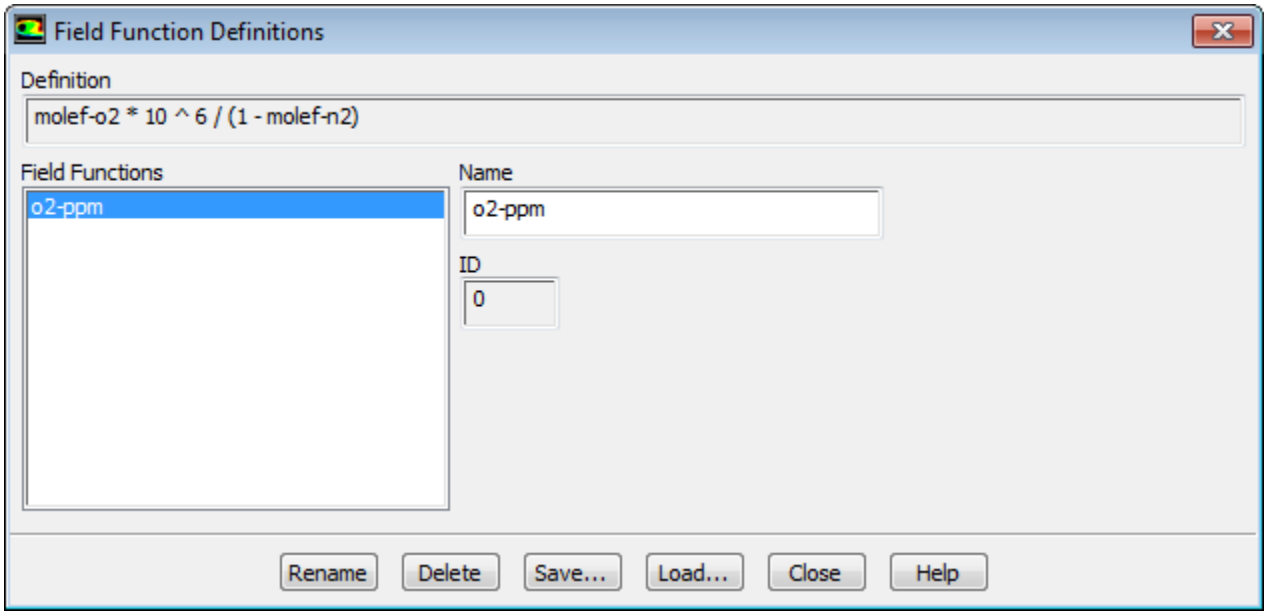
Create Output Parameter
creates an output parameter for this report definition with the name `<report definition name>-op`.

Define
creates the report definition.

38.2.30. Field Function Definitions Dialog Box

The **Field Function Definitions** dialog box allows you to check, rename, save, load, and delete custom field functions that you defined in the Custom Field Function Calculator Dialog Box (p. 2683). See Manipulating, Saving, and Loading Custom Field Functions (p. 2006) for details about the items below.

The **Field Function Definitions** dialog box is opened from the Custom Field Function Calculator Dialog Box (p. 2683).



Controls

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Definition
displays the function selected in the **Field Functions** list. This display is for informational purposes only; you cannot edit it.

Field Functions
contains a selectable list of custom field functions. When you select a function, its definition will appear in the **Definition** box and its name will appear in the **Name** text entry box.

Name
displays the name of the currently selected field function. You can enter a new name in this box if you want to rename the function.

ID
reports the ID number of the selected function. The field function at the top of the list has an ID of 0, the second function has an ID of 1, and so on.

Rename
changes the name of the selected function to the name specified in the **Name** text entry box.

Delete
deletes the selected field function.

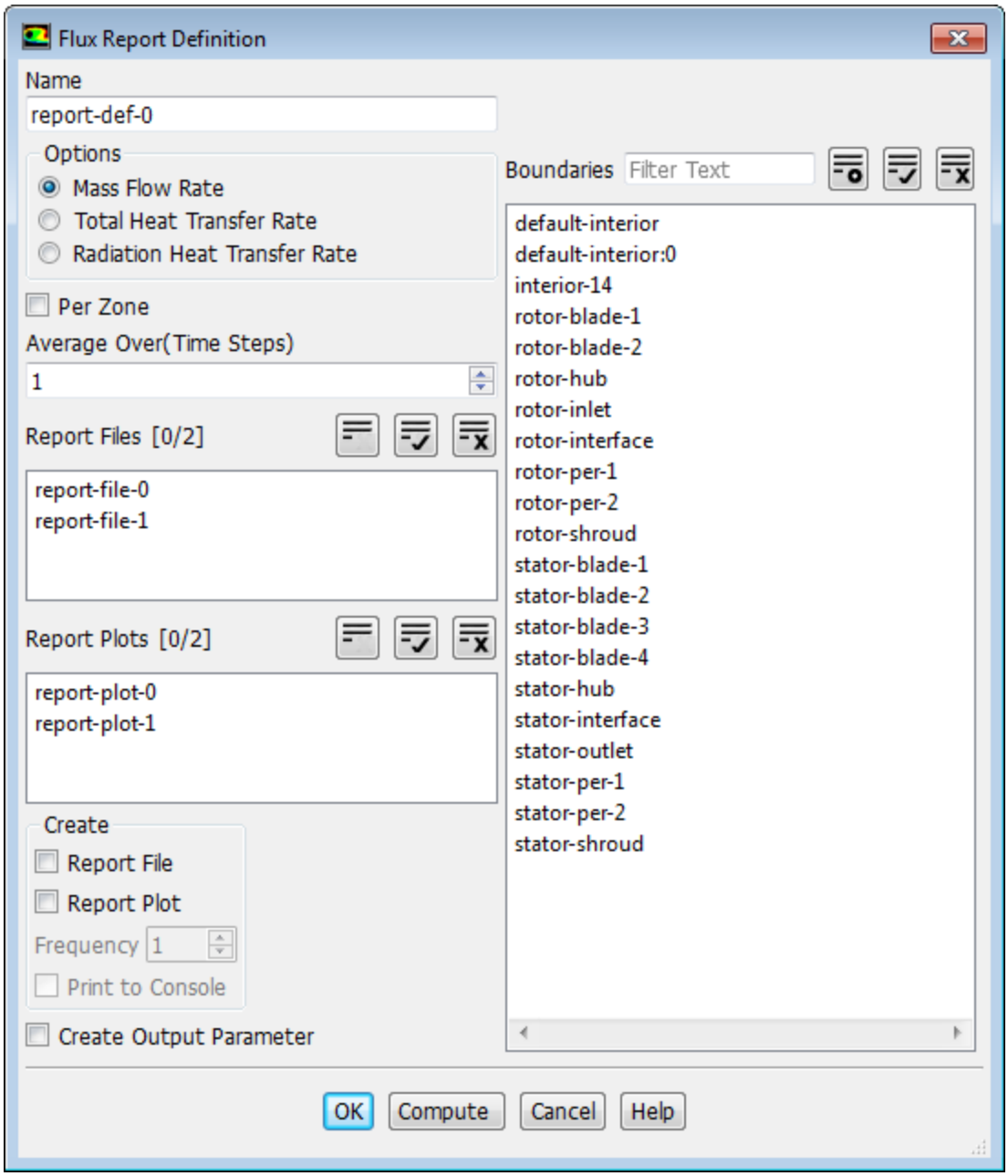
Save...
opens The Select File Dialog Box (p. 19), where you can specify a file in which to save all of the functions in the **Field Functions** list.

Load...
opens the **Select File** dialog box, where you can specify a file from which to read custom field functions (that is, a file that you saved using the **Save...** button above).

38.2.31. Flux Report Definition Dialog Box

Solving → Reports → Definitions → New → Flux Report

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options

Mass Flow Rate
turns on the computation of the mass flow rate for the selected boundary zones.

Total Heat Transfer Rate
turns on the computation of the total heat transfer rate for the selected boundary zones.

Total Sensible Heat Transfer Rate
turns on the computation of the total sensible heat transfer rate for the selected boundary zones. It reports the total energy flux as defined in Equation 5.2 in the Theory Guide.

Radiation Heat Transfer Rate
turns on the computation of the radiation heat transfer rate for the selected boundary zones.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Film Mass Flow Rate
(available only when the Eulerian wall film model is enabled) turns on the computation of the mass flow rate for the selected boundary zone(s).

Film Heat Transfer Rate
(available only when the Eulerian wall film model is enabled) turns on the computation of the film heat transfer rate for the selected boundary zone(s).

Phase
contains a list of all of the phases in the problem that you have defined. This is available when the VOF, mixture, or Eulerian multiphase model is enabled.

Per Zone
specifies whether or not the chosen field variable is calculated from all of the selected boundary zones combined (default) or individually on each of the selected boundary zones.

Average Over (Time-Steps), Average Over (Iterations)
(optional) To have Fluent calculate a running average for the **Flux Report Definition** you can enter a positive integer greater than 1 (the default) for **Average Over**.

Specifying a number greater than 1 means that Fluent will print, plot, and write the running average value of the selected variable instead of the current value of the same variable.

The value reported is averaged over the last N *iterations/time steps, where* N *is your specified* ***Average Over*** *value. When the iteration/time step number is lower than* N*, Fluent calculates the average of the available variable values.*

Report Files
lists all of the report files where you can write the report definition data.

Report Plots
lists all of the report plots where you can plot the report definition data (assuming this report has the same units as the other report definitions included in the selected report plot).

Create

Report File
creates a new report file that includes this report definition.

Report Plot
creates a new report plot that includes this report definition.

Frequency
specifies how frequently the report definition is written, plotted, or printed to the console.

Print to Console
prints the value of the report definition to the console.

Create Output Parameter
creates an output parameter for this report definition with the name `<report definition name>-op`.

Report Type
selects the integration method used on the selected cell zones. The available report types are the same as those in the Volume Integrals Dialog Box (p. 2620). See Volume Integration (p. 1922) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Boundaries
contains a selectable list of valid boundary zones for flux reporting.

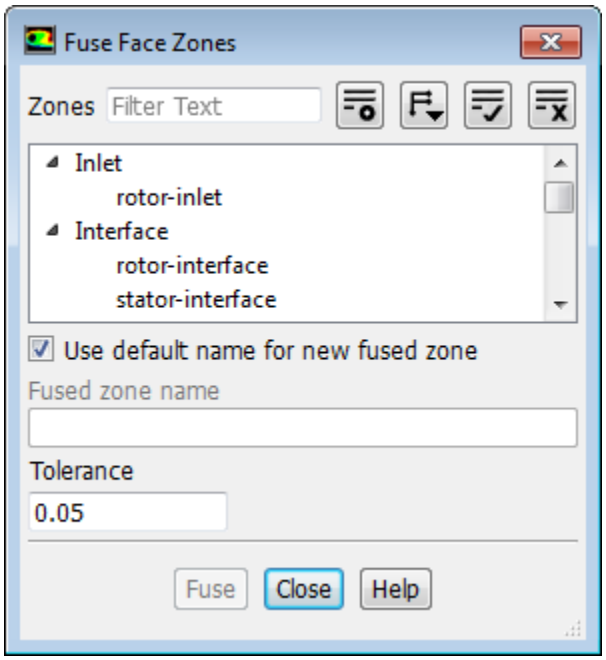
OK
creates the report definition.

Compute
computes the value of the report definition at the selected zone(s) and prints to the console.

38.2.32. Fuse Face Zones Dialog Box

The **Fuse Face Zones** dialog box allows you to fuse boundaries (that is, remove duplicate nodes and faces and delete artificial internal boundaries) created by assembling multiple mesh regions. (See Reading Multiple Mesh/Case/Data Files (p. 145) for details on importing such meshes.) See Fusing Face Zones (p. 212) for information about using this dialog box.

Setting Up Domain → Zones → Combine... → Fuse...



Controls

Zones
contains a list of face zones from which you can select the boundaries to be fused.

Use default name for new fused zone
specifies whether to use an automatically generated default name for the fused zone.

Fused zone name
specifies the desired name for the fused zone if **Use default name for new fused zone** is disabled.

Tolerance
is a fraction of the minimum edge length of the face, used to determine whether or not nodes are coincident. If all of the appropriate faces do not get fused using the default **Tolerance**, you should increase it and attempt to fuse the zones again. The **Tolerance** should not exceed 0.5, or you may fuse the wrong nodes.

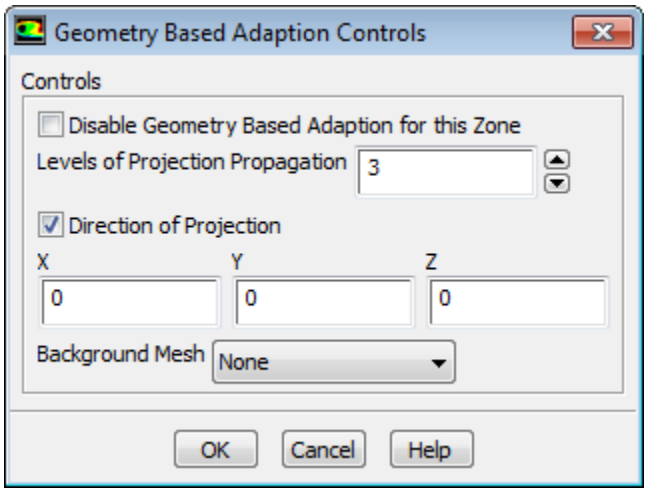
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fuse
fuses the selected zones. It is enabled only when you have selected a minimum of two zones.

38.2.33. Geometry Based Adaption Controls Dialog Box

The **Geometry Based Adaption Controls** dialog box allows you to set projections. See Performing Geometry-Based Adaption (p. 1699) for details about the items below.

The **Geometry Based Adaption Controls** dialog box is opened from the Geometry Based Adaption Dialog Box (p. 2709).



Controls

Controls
contains parameters to define geometry based adaption.

Disable Geometry Based Adaption for this Zone
disables the geometry reconstruction for selected zones in the domain.

Levels of Projection Propagation
is the number of layers of the nodes you want to project.

Direction of Projection
allows you to specify the direction in which you want to project the nodes.

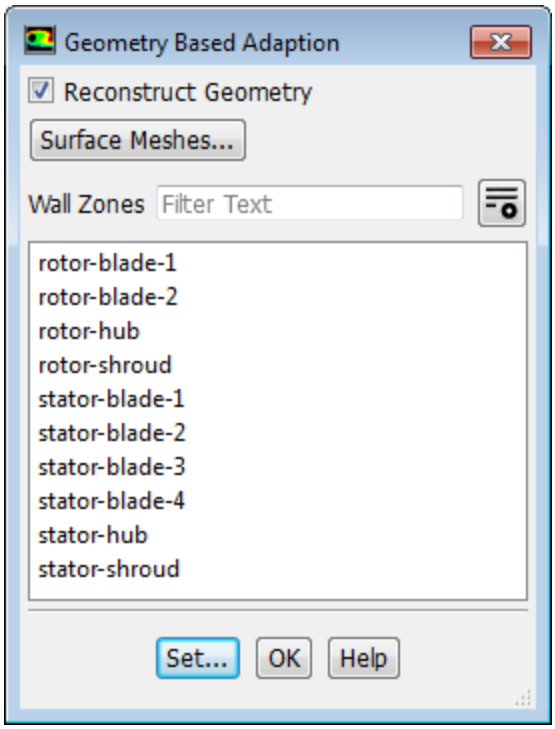
Background Mesh
allows you to load the surface mesh as a background mesh for adaption.

38.2.34. Geometry Based Adaption Dialog Box

The **Geometry Based Adaption** dialog box allows you to reconstruct the geometry while performing boundary adaption. See Geometry-Based Adaption (p. 1698) for details.

Setting Up Domain → Adapt → More → Geometry...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Reconstruct Geometry
enables geometry based adaption parameters.

Surface Meshes...
opens the Surface Meshes Dialog Box (p. 2807).

Wall Zones
allows you to select the wall zones you want to adapt.

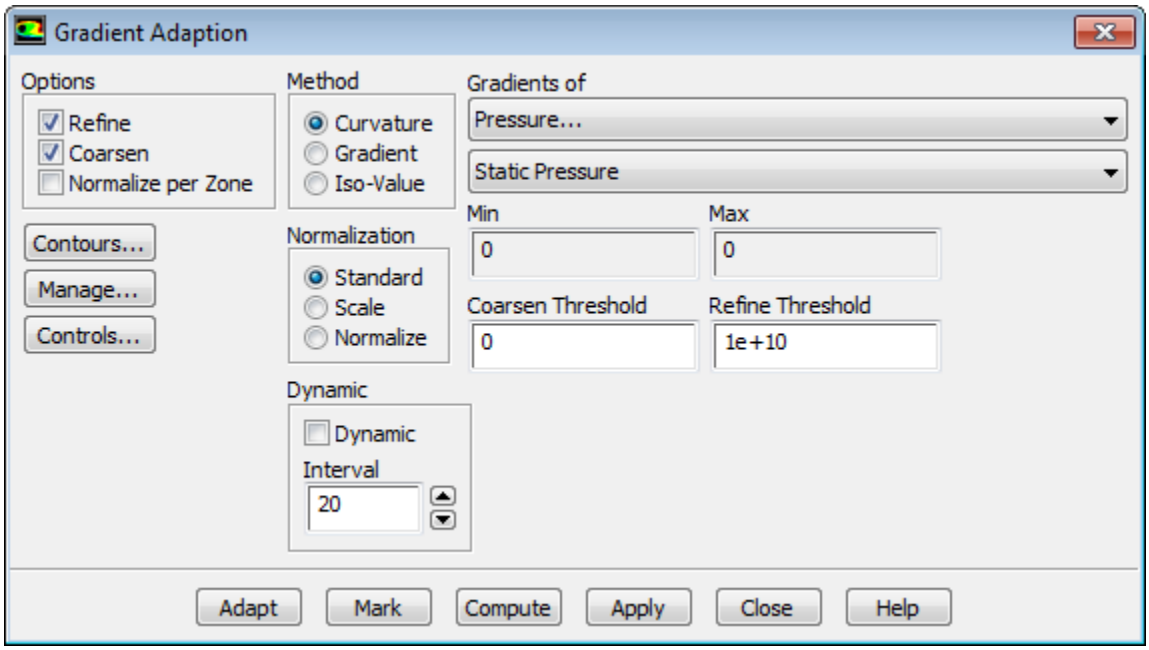
Set...
opens the Geometry Based Adaption Controls Dialog Box (p. 2709).

38.2.35. Gradient Adaption Dialog Box

The **Gradient Adaption** dialog box allows you to mark or adapt to gradients of a specified field variable. See Gradient Adaption (p. 1688) for details.

Setting Up Domain → Adapt → Mark/Adapt Cells → Gradient...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options
contains the check buttons that toggle the ability to mark and/or adapt cells for refinement or coarsening.

Refine
toggles the ability to refine or mark cells for refinement.

Coarsen
toggles the ability to coarsen or mark cells for coarsening (available in 2D and axisymmetric cases).

Normalize per Zone
toggles the ability of zonal normalization.

Contours...
opens the Contours Dialog Box (p. 2676), which you can use as an aid to selecting adaption thresholds by displaying contours of adaption function.

Manage...
opens the Manage Adaption Registers Dialog Box (p. 2728), which allows you to display and manipulate adaption registers that are generated using the **Mark** command.

Controls...
opens the Mesh Adaption Controls Dialog Box (p. 2730), which allows you to control certain aspects of the adaption process.

Method
contains options for specifying the criterion for adaption.

Curvature
specifies the use of the second gradient of a field variable for adaption. This approach is recommended for problems with smooth solutions.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Gradient
specifies the use of the first gradient of a field variable for adaption. This approach is recommended for problems with strong shocks.

Iso-Value
allows you to customize the adaption criterion (using custom field functions, user-defined scalars, and so on).

Normalization
contains the options available for normalization.

Standard
specifies that the gradient or curvature is not normalized.

Scale
specifies that the gradient or curvature is scaled by its average value in the domain.

Normalize
specifies that the gradient or curvature is scaled by its maximum value in the domain (that is, the gradient or curvature is bounded by [0, 1]).

Dynamic
contains options to specify dynamic gradient adaption.

Dynamic
enables dynamic gradient adaption.

Interval
allows you to specify the number of iterations or time-steps between two consecutive automatic mesh adaptions, depending on whether you are performing a steady-state or a time-dependent solution, and on which solver you are using.

Gradients of
contains a list of the field variables that can be used in the gradient adaption function.

Min/Max
displays the minimum and maximum cell values of the gradient adaption function based on the selected quantity.

Coarsen Threshold
designates the threshold values for coarsening the mesh. Cells with adaption function values below the **Coarsen Threshold** will be marked for coarsening.

Refine Threshold
designates the threshold values for refining the mesh. Cells with adaption function values above the **Refine Threshold** will be marked for refinement.

Adapt
adapts the mesh based on the gradients of the selected scalar quantity, the coarsening and refining toggle buttons and thresholds, and the adaption limits.

Mark
marks cells to be refined and/or coarsened based on the gradients of the selected quantity and the coarsening and refining toggle buttons and thresholds. This command produces an adaption register. For information on using adaption registers, see Manipulating Adaption Registers (p. 1701).

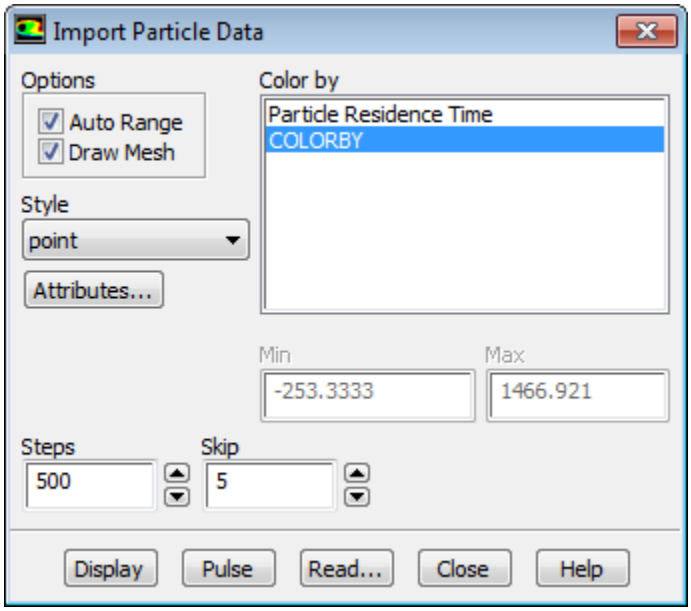
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Compute
computes the minimum and maximum cell values of the gradient adaption function with the selected scalar quantity. The values are displayed in the **Min** and **Max** real number fields.

38.2.36. Import Particle Data Dialog Box

The **Import Particle Data** dialog box allows you to import particle history data for display purposes. See Importing Particle Data (p. 1338) for details.

Postprocessing → Model Specific → Discrete Phase → Import Particle Data...



Controls

Options
contains the check buttons that set various import particle data options.

Auto Range
toggles between automatic and manual setting of the particle data range.

Draw Mesh
toggles between displaying and not displaying the mesh. The Mesh Display Dialog Box (p. 2163) is opened when **Draw Mesh** is selected.

Style
allows you to select pathline style.

Attributes...
opens the Path Style Attributes Dialog Box (p. 2559). This allows you to modify the line width, cylinder radius or marker size.

Color by
contains a list from which you can select the scalar field to be used to color the particle data.

Min,Max
allows you to set the range when **Auto Range** is disabled.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Steps
sets the maximum number of steps a particle can advance.

Skip
allows you to "thin out" the pathlines.

Display
displays pathlines.

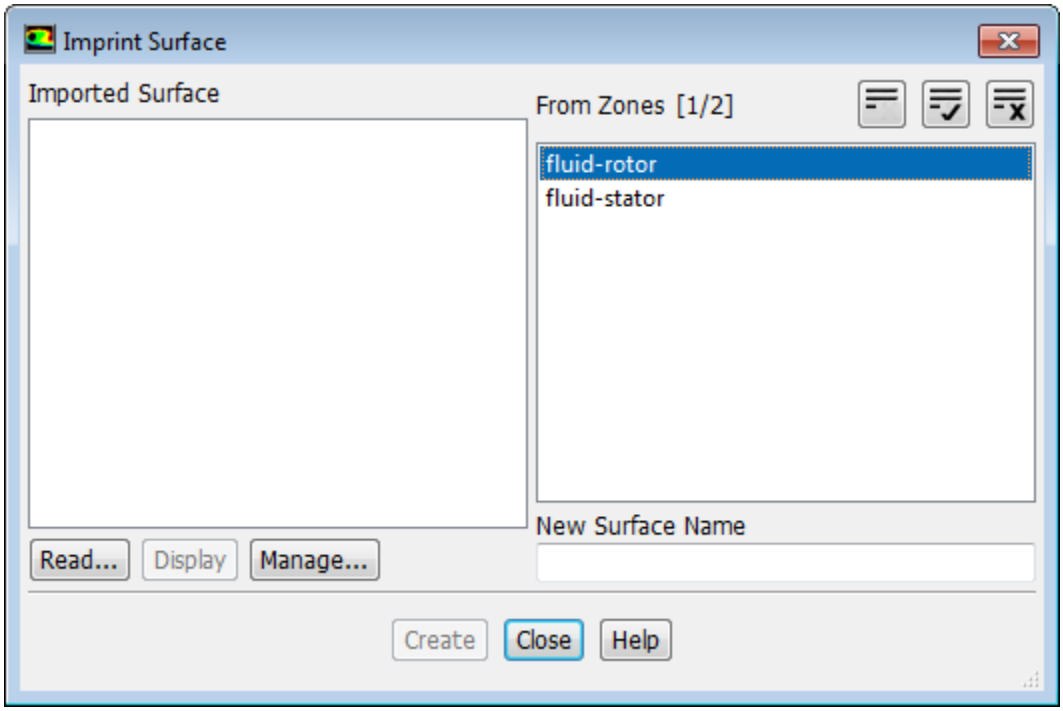
Pulse
animates the particle positions. The **Pulse** button will become the **Stop !** button during the animation, and you must click **Stop !** to stop the pulsing.

Read...
opens a file selection dialog box where you can enter a file name and a directory for the imported data.

38.2.37. Imprint Surface Dialog Box

The **Imprint Surface** dialog box allows you to define a surface on top of an existing cell zone for postprocessing purposes. See Imprint Surfaces (p. 1714) for details about the following items.

Setting Up Domain → Surface → Create → Imprint...



Controls

Imported Surface
lists the surfaces that you have imported.

Read...
opens The Select File Dialog Box (p. 19), so that you can select surface files for import.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Display
allows you to display the selected surfaces.

Manage...
opens the Surfaces Dialog Box (p. 2810), where you can rename and delete surfaces and determine their sizes.

New Surface Name
designates the name of the new surface. The default name format is `<original-name>(imprint)-<id>`.

From Zones
allows you to select the zones where you want to import/imprint the surface mesh.

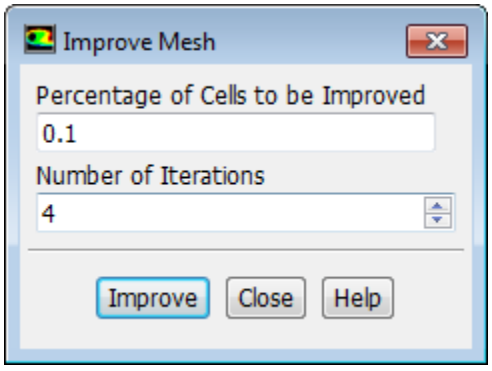
> **Note**
>
> If no zone is selected, then the entire domain is considered.

Create
creates the new surface, imprinted with the mesh of the selected cell zone.

38.2.38. Improve Mesh Dialog Box

The **Improve Mesh** dialog box controls the quality-based smoothing of the numerical mesh. See Improving the Mesh by Smoothing and Swapping (p. 230) for details about the items below.

Setting Up Domain → Mesh → Improve...



Controls

Percentage of Cells to be Improved
sets the percentage of the total cells on which improvements are attempted. Note that the cells selected for improvement are in the areas of the mesh that exhibit the lowest orthogonal quality.

Number of Iterations
defines the number of successive smoothing sweeps performed on the mesh.

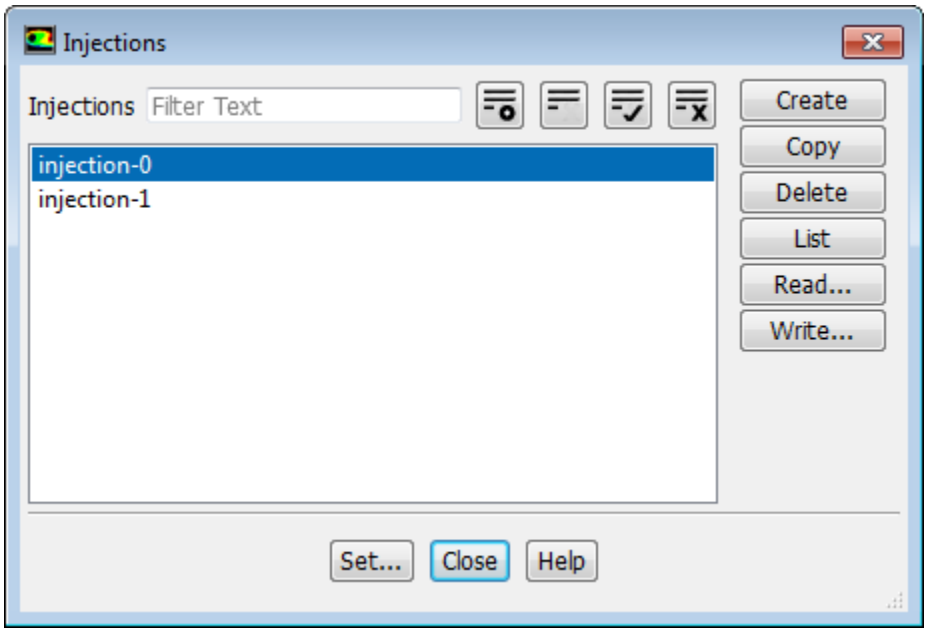
Improve
initiates the desired number of smoothing iterations.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.2.39. Injections Dialog Box

The **Injections** dialog box allows you to create, delete, and list discrete phase injections, and access the Set Injection Properties Dialog Box (p. 2796) and the Set Multiple Injection Properties Dialog Box (p. 2803), in which you can set the properties for the injections. See Creating and Modifying Injections (p. 1284) for details.

Setting Up Physics → Model Specific → Discrete Phase → Injections...



Controls

Injections
contains a list from which you can select one or more injections in order to set, copy, or modify properties, or delete or list injections.

Create
creates a new injection and opens the Set Injection Properties Dialog Box (p. 2796), in which you can set its properties.

Copy
creates a new injection with the same properties as the selected injection and opens the Set Injection Properties Dialog Box (p. 2796) where the new injection's properties can be modified.

Delete
deletes the injection(s) selected in the **Injections** list.

List
lists the initial conditions for the particle streams in the injection(s) selected in the **Injections** list.

Read...
opens The Select File Dialog Box (p. 19) where you will select the injection file to read in.

Write...
allows you to select the injection from the list and write it to a file.

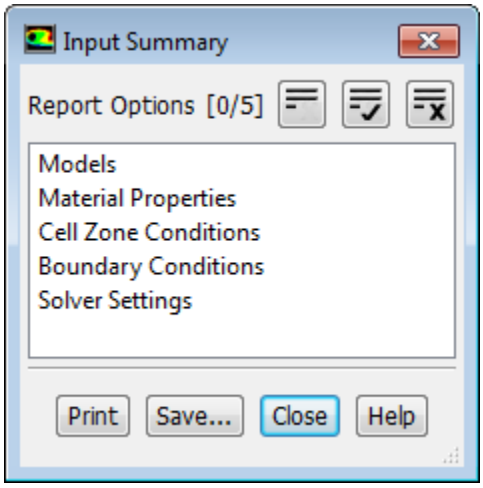
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Set...
opens the Set Injection Properties Dialog Box (p. 2796) for the injection selected in the **Injections** list or the Set Multiple Injection Properties Dialog Box (p. 2803) if more than one injection is selected in the **Injections** list.

38.2.40. Input Summary Dialog Box

The **Input Summary** dialog box allows you to report the current settings for physical models, boundary conditions, material properties, and solution parameters. See Generating a Summary Report (p. 1927) for details about the items below.

Solving → Run Calculation → Input Summary...



Controls

Report Options
contains a selectable list of the information that is available for the report.

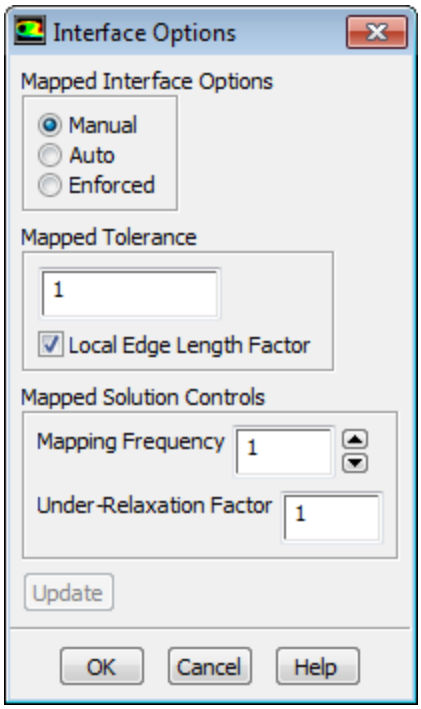
Print
prints the selected information to the console.

Save...
opens The Select File Dialog Box (p. 19), in which you can specify the filename under which to save the output of the summary report.

38.2.41. Interface Options Dialog Box

The **Interface Options** dialog box allows you to update which mesh interfaces use the mapped option and the related settings (see The Mapped Option (p. 158) for additional information). It is opened by clicking the **Options...** button in the **Mesh Interfaces** group box of the Mesh Interfaces Dialog Box (p. 2732).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Mapped Interface Options
contains options related to mapped interface specification.

Manual
specifies that the only **Mapped** mesh interfaces are the ones that you have defined as **Mapped** in the **Edit Mesh Interfaces** dialog box.

Auto
specifies that all mesh interfaces that have interface zones that penetrate each other are automatically defined as **Mapped** interfaces.

Enforced
specifies all mesh interfaces as **Mapped** interfaces. Fluent creates the mapped interfaces regardless of the mesh quality.

Mapped Tolerance

Local Edge Length Factor
creates mapping based on local mesh size. The default factor is **1**. Disabling **Local Edge Length Factor** allows you to specify an absolute value for the tolerance that is used in determining the search range for mapping. Note that this does not overwrite any locally specified tolerance in the **Edit Mesh Interfaces** dialog box.

Mapped Solution Controls

Mapping Frequency
specifies how often calculation values are interpolated across mapped interfaces.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Under-Relaxation Factor
allows you to reduce the rate of change per iteration of physical quantities across mapped interfaces to aid with convergence. For additional information on setting under-relaxation factors, see Setting Under-Relaxation Factors (p. 1564).

Update
applies the selection(s) made in the **Interface Options** dialog box. This button is grayed-out until you make a selection in the **Mapped Interface Options** or **Mapped Tolerance** group boxes; it also returns to being grayed-out once it is clicked.

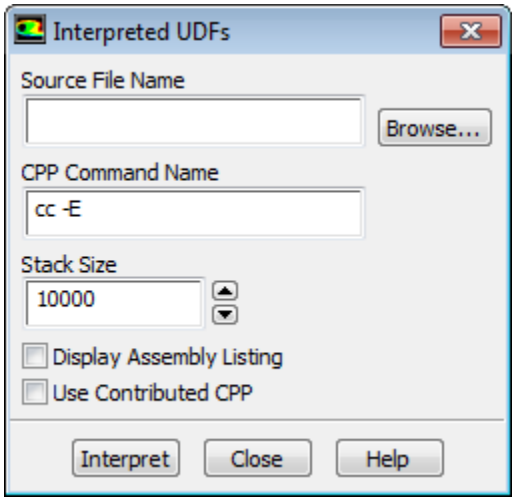
Note

Selecting **Auto** or **Enforced** and clicking **Update** applies these selections globally for all relevant interfaces. Once you click **Update** you can return to the **Mesh Interfaces** dialog box to modify local settings for individual interfaces.

38.2.42. Interpreted UDFs Dialog Box

The **Interpreted UDFs** dialog box allows you to compile user-defined functions. See the separate Fluent Customization Manual for details.

User Defined → **User Defined** → **Functions** → **Interpreted...**



Controls

Source File Name
sets the name of your user-defined function.

CPP Command Name
sets the name of your C preprocessor.

Stack Size
sets the size of the stack. Keep the default **Stack Size** setting of 10000, unless the number of local variables in your function will cause the stack to overflow. In this case, set the **Stack Size** to a number that is greater than the number of local variables used.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

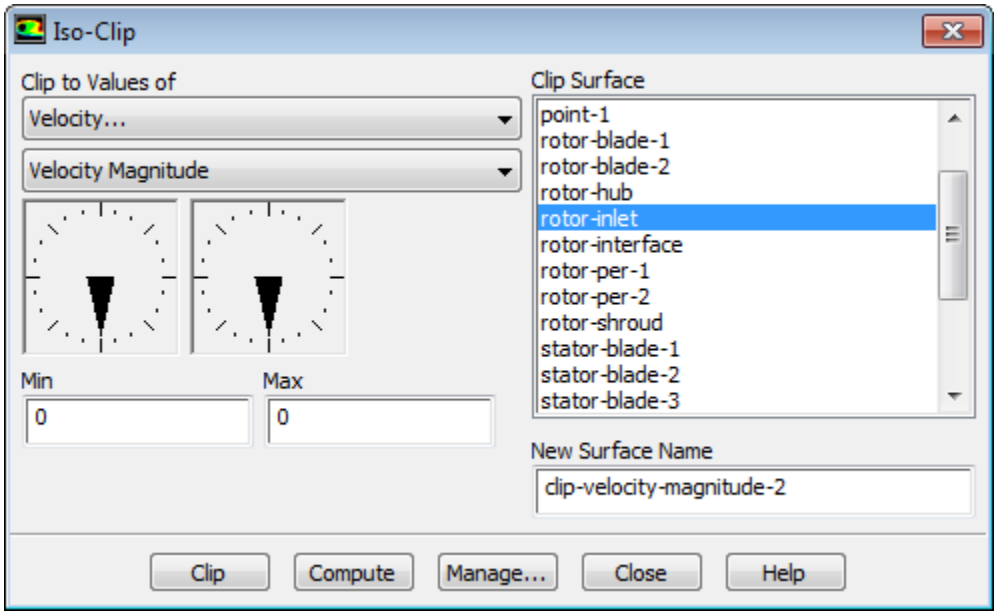
Display Assembly Listing
indicates whether or not to display a listing of assembly language code in your console window as the function compiles.

Use Contributed CPP
specifies the use of the C preprocessor that ANSYS Fluent has supplied, instead of your own.

Interpret
interprets the specified function.

38.2.43. Iso-Clip Dialog Box

The **Iso-Clip** dialog box allows you to interactively clip surfaces. The clipped surface consists of those points on the selected surface where the scalar field values are within the specified range. See Clipping Surfaces (p. 1730) for details about the items below.



Controls

Clip to Values of
contains a list from which you can select the scalar field to be used for clipping.

Phase
allows you to select the phase when one of the multiphase models is selected in the Multiphase Model Dialog Box (p. 2171).

Min, Max
set the minimum and maximum values in the clipping range. There are two ways you can set the **Min** and **Max**:

- You can set a value interactively by moving the indicator in the dial above the **Min** or **Max** field with the left mouse button. This will also create a temporary surface in the graphics window. Using the dial allows you to preview the clipped surfaces before defining them. Note: Even though the clipped surface is displayed, it is only a temporary surface. To clip the surface, use the **Clip** button after deciding on the minimum and maximum values.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You can type in values in the **Min** and **Max** fields directly.

Clip Surface
contains a list of surfaces from which you can select the surface to be clipped. You must select a surface to make the **Clip** button available.

New Surface Name
designates the name of the surface to be created. The default is the concatenation of the surface type and an integer which is the new surface ID.

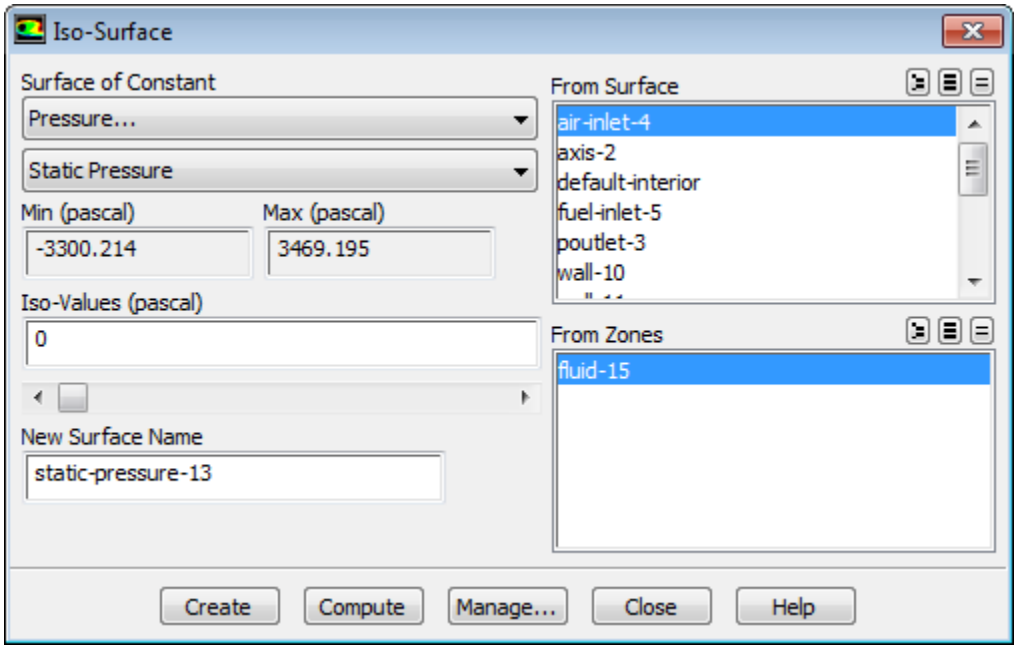
Clip
creates the clipped surface. (The original surface will remain unchanged.)

Compute
computes the minimum and maximum values of the scalar field across the domain, and displays them in the **Min** and **Max** boxes.

Manage...
opens the Surfaces Dialog Box (p. 2810) in which you can rename and delete surfaces and determine their sizes.

38.2.44. Iso-Surface Dialog Box

The **Iso-Surface** dialog box allows you to interactively create isosurfaces. These surfaces can be isovalued sections of an existing surface or of the entire domain. For more effective use of the slider bar, press the **Compute** button before using it. See Isosurfaces (p. 1728) for details about the items below.



Controls

Surface of Constant
contains a list from which you can select the scalar field which will be used for isosurfacing.

Min
displays the minimum field value, which is computed when you click **Compute**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Max
displays the maximum field value, which is computed when you click **Compute**.

Iso-Values
sets user-specified isovalues. There are two ways you can set the **Iso-Values**:

- You can set an isovalue interactively by moving the slider with the left mouse button. This will also create a temporary isosurface in the graphics window. Using the slider allows you to preview the isosurfaces before defining them. Note: Even though the isosurface is displayed, it is only a temporary surface. To create an isosurface, use the **Create** button after deciding on a particular isovalue.
- You can type in isovalues in the **Iso-Values** field directly, separating multiple values by white space. Multiple isovalues will be contained in a single isosurface; that is, you cannot select subsurfaces within the resulting isosurface.

From Surface
contains a list of existing surfaces from which you can select the surface to be used for isosurfacing. If you do not select a surface from the list, the isosurfacing will be performed on the entire domain.

From Zones
contains a list of cell zones from which you can select the zone for creating an isosurface.

New Surface Name
designates the name of the surface to be created. The default is the concatenation of the scalar field name and an integer which is the new surface ID.

Create
creates the surface.

Compute
computes the minimum and maximum of the scalar field across the domain and displays them in the **Min** and **Max** boxes.

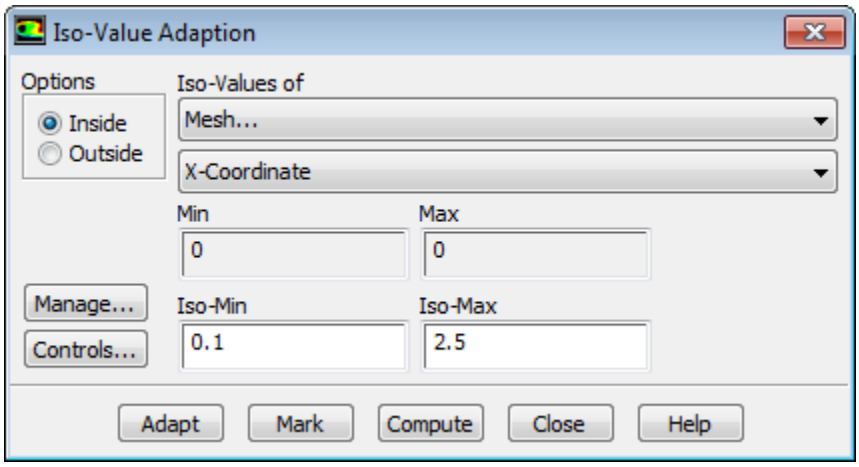
Manage...
opens the Surfaces Dialog Box (p. 2810) in which you can rename and delete surfaces and determine their sizes.

38.2.45. Iso-Value Adaption Dialog Box

The **Iso-Value Adaption** dialog box allows you to mark or refine cells inside or outside a specified range of a selected scalar function. See Isovalue Adaption (p. 1691) for details.

Setting Up Domain → Adapt → Mark/Adapt Cells → Iso-Value...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options
contains radio buttons that control whether the cells inside or outside the isovalue range are marked for refinement.

Inside
enables the marking of cells with values between **Iso-Min** and **Iso-Max**.

Outside
enables the marking of cells with values less than **Iso-Min** or greater than **Iso-Max**.

Iso-Values of
contains a list from which you can select the solution variable to be used in the isovalue adaption function.

Min/Max
displays the minimum and maximum cell values of the selected field variable. The real number field values are not editable; they are purely informational.

Iso-Min
defines the minimum isovalue threshold.

Iso-Max
defines the maximum isovalue threshold.

Manage...
opens the Manage Adaption Registers Dialog Box (p. 2728), which allows you to display and manipulate adaption registers that are generated using the **Mark** command.

Controls...
opens the Mesh Adaption Controls Dialog Box (p. 2730), which allows you to control certain aspects of the adaption process.

Adapt
adapts the mesh based on the isovalues of the selected solution variable, the isovalue ranges, and the **Inside/Outside** option.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mark

marks cells to be refined based on the isovalues of the selected quantity, the isovalue ranges, and the **Inside/Outside** option. This command produces an adaption register. For information on using adaption registers, see Manipulating Adaption Registers (p. 1701).

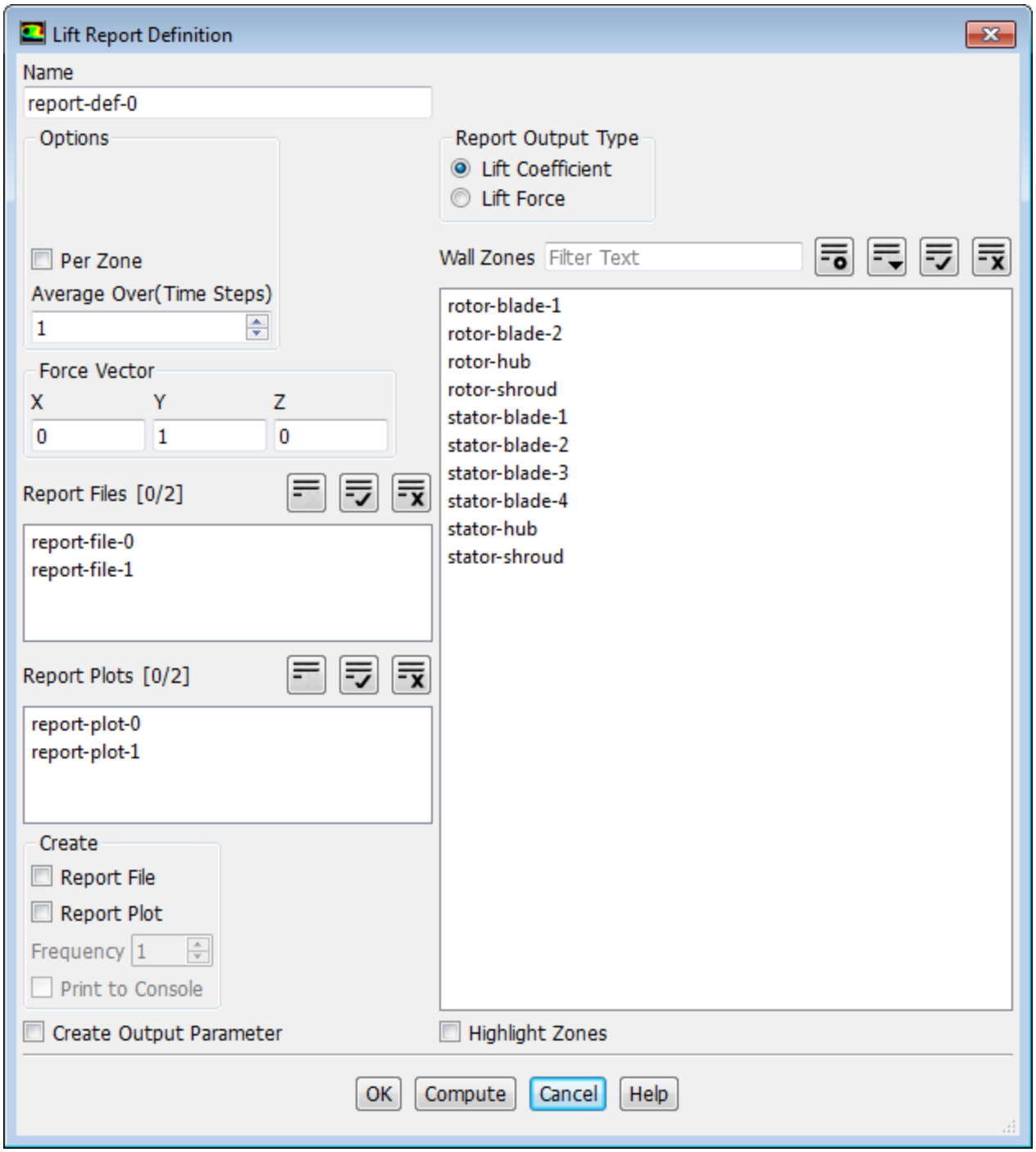
Compute

computes the minimum and maximum cell values of the selected solution variable and displays them in the **Min** and **Max** real number fields.

38.2.46. Lift Report Definition Dialog Box

Solving → **Reports** → **Definitions** → **New** → **Force Report** → **Lift...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options

Per Zone

specifies whether or not the lift for multiple walls zones should be monitored on each wall zone separately.When this option is turned off (the default), Fluent computes and monitors the total lift for all of the selected walls zones combined together.

Average Over (Time-Steps), Average Over (Iterations)

(optional) To have Fluent calculate a running average for the **Lift Report Definition** you can enter a positive integer greater than 1 (the default) for **Average Over**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specifying a number greater than 1 means that Fluent will print, plot, and write the running average value of the selected variable instead of the current value of the same variable.

The value reported is averaged over the last N *iterations/time steps, where* N *is your specified* ***Average Over*** *value. When the iteration/time step number is lower than* N*, Fluent calculates the average of the available variable values.*

Force Vector
contains the components of the force vector.

X,Y,Z
are the components of the force vector along which the forces will be computed.

Report Files
lists all of the report files where you can write the report definition data.

Report Plots
lists all of the report plots where you can plot the report definition data (assuming this report has the same units as the other report definitions included in the selected report plot).

Create

Report File
creates a new report file that includes this report definition.

Report Plot
creates a new report plot that includes this report definition.

Frequency
specifies how frequently the report definition is written, plotted, or printed to the console.

Print to Console
prints the value of the report definition to the console.

Create Output Parameter
creates an output parameter for this report definition with the name `<report definition name>-op`.

Report Output Type

Lift Coefficient
specifies the report definition as a unitless quantity.

Lift Force
specifies that the report definition has units of force.

Wall Zones
lists the zones you can select for lift coefficient or drag lift computation.

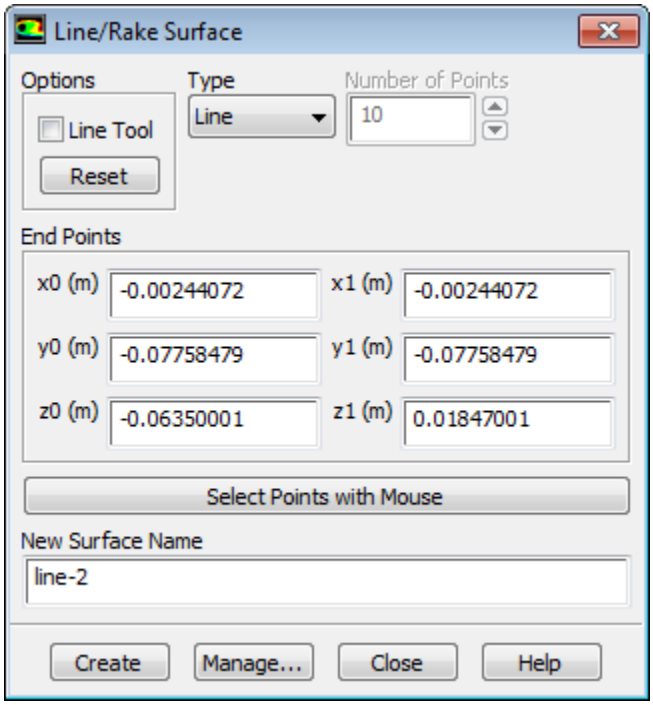
OK
creates the report definition.

Compute
computes the value of the report definition at the selected zone(s) and prints to the console.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.2.47. Line/Rake Surface Dialog Box

The **Line/Rake Surface** dialog box allows you to interactively create line and rake surfaces. A rake is a linear, uniform distribution of points between two endpoints. See Line and Rake Surfaces (p. 1718) for details about the items below.



Controls

Options
contains options related to the line tool. See Using the Line Tool (p. 1720) for details about using this feature.

Line Tool
enables the line tool.

Reset
resets the line tool to its default position.

Type
selects line or rake as the surface to be created.

Number of Points
defines the number of points in the rake surface (inactive for line surfaces).

End Points
designates the coordinates of the first point (**x0**, **y0**, **z0**) and the last point (**x1**, **y1**, **z1**).

Select Points with Mouse
allows you to select endpoints with the mouse. You can select endpoints by clicking on locations in the active window with the mouse-probe button. (See Controlling the Mouse Button Functions (p. 1800) for information about setting mouse button functions.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

New Surface Name
designates the name of the new surface. The default is the concatenation of the surface type and an integer which is the new surface ID.

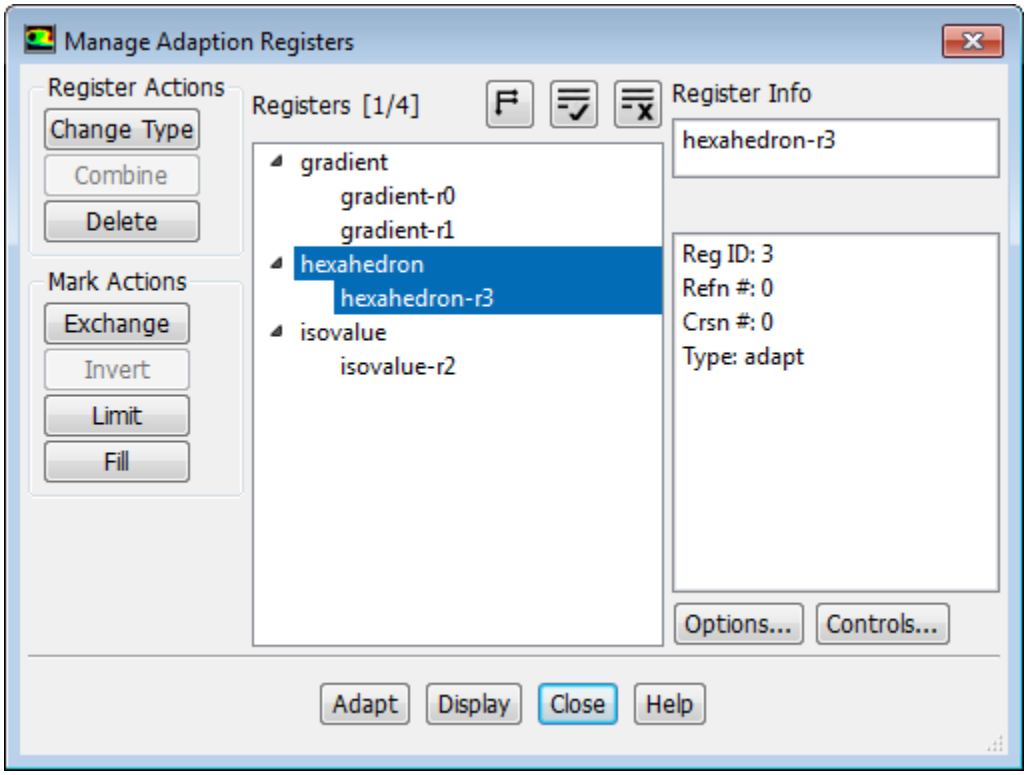
Create
creates the surface.

Manage...
opens the Surfaces Dialog Box (p. 2810) in which you can rename and delete surfaces and determine their sizes.

38.2.48. Manage Adaption Registers Dialog Box

The **Manage Adaption Registers** dialog box provides an interactive mechanism for creating, destroying, and displaying functions for mesh adaption. See Manipulating Adaption Registers (p. 1701) for details about using the items in this dialog box.

Setting Up Domain → Adapt → Manage Registers...



Controls

Register Actions
contains operations applied to adaption or mask registers.

Change Type
toggles the register between adaption and mask types. Adaption registers are used to initiate refining or coarsening of the mesh. Typically, mask registers are combined with adaption registers to control the scope of the adaption process.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Combine
combines the selected adaption registers to create a hybrid adaption function. In some instances, three new registers may be created: a combination of the adaption registers, a combination of the mask registers, and then a combination of the two combined registers.

Delete
permanently discards the selected registers.

Mark Actions
contains operations applied to the cell markings defined in an adaption or mask register.

Exchange
modifies the cell markings in the following manner: all cells originally marked for refinement are marked for coarsening, and all cells originally marked for coarsening are marked for refinement.

Invert
modifies all the cell markings in a mask register in the following manner: all cells that were originally marked as ACTIVE are marked INACTIVE, and all cells originally marked as INACTIVE are marked ACTIVE. Note that this action can only be applied to mask registers.

Limit
applies the adaption volume limits to the selected registers. For information on adaption limits, see Mesh Adaption Controls (p. 1705).

Fill
marks for coarsening all cells in the adaption register that are not marked for refinement.

Registers
contains a list from which you can select the current adaption and mask registers. The availability of many of the actions in the dialog box is based on the number and/or type of adaption registers selected.

Register Info
provides the name, ID, number of cells marked for refinement and coarsening, and the type of the most recently selected or deselected register.

Options...
opens the Adaption Display Options Dialog Box (p. 2663).

Controls...
opens the Mesh Adaption Controls Dialog Box (p. 2730), which allows you to control certain aspects of the adaption process.

Adapt
adapts the mesh based on the selected adaption register. The **Adapt** button is not available if more than one adaption register is selected. Adaption functions composed of combinations of adaption registers can be produced using commands in the **Register Actions** and **Mark Actions** boxes.

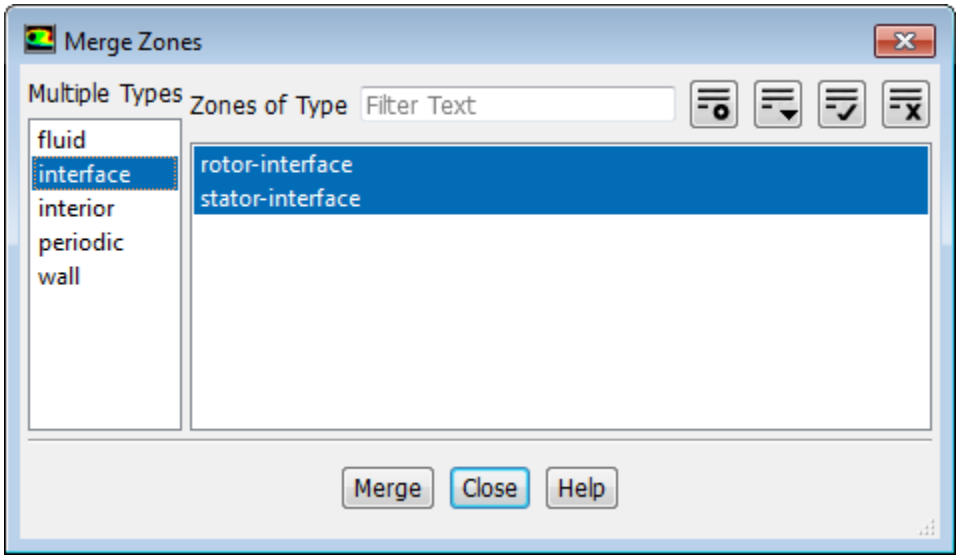
Display
displays the cells marked for adaption in the selected adaption register in the active graphics window. The **Display** button is not available if more than one adaption register is selected. Adaption functions composed of combinations of adaption registers can be produced using commands in the **Register Actions** and **Mark Actions** boxes.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.2.49. Merge Zones Dialog Box

The **Merge Zones** dialog box allows you to merge multiple zones of the same type into a single zone. See Merging Zones (p. 206) for details.

Setting Up Domain → Zones → Combine → Merge...



Controls

Multiple Types
contains a selectable list of the zone types for which you can merge multiple zones. You select a type and the corresponding zones of that type will be displayed in the **Zones of Type** list.

Zones of Type
contains a list from which you can select two or more zones to be merged into a single zone. The zones are of the type selected in the **Multiple Types** list.

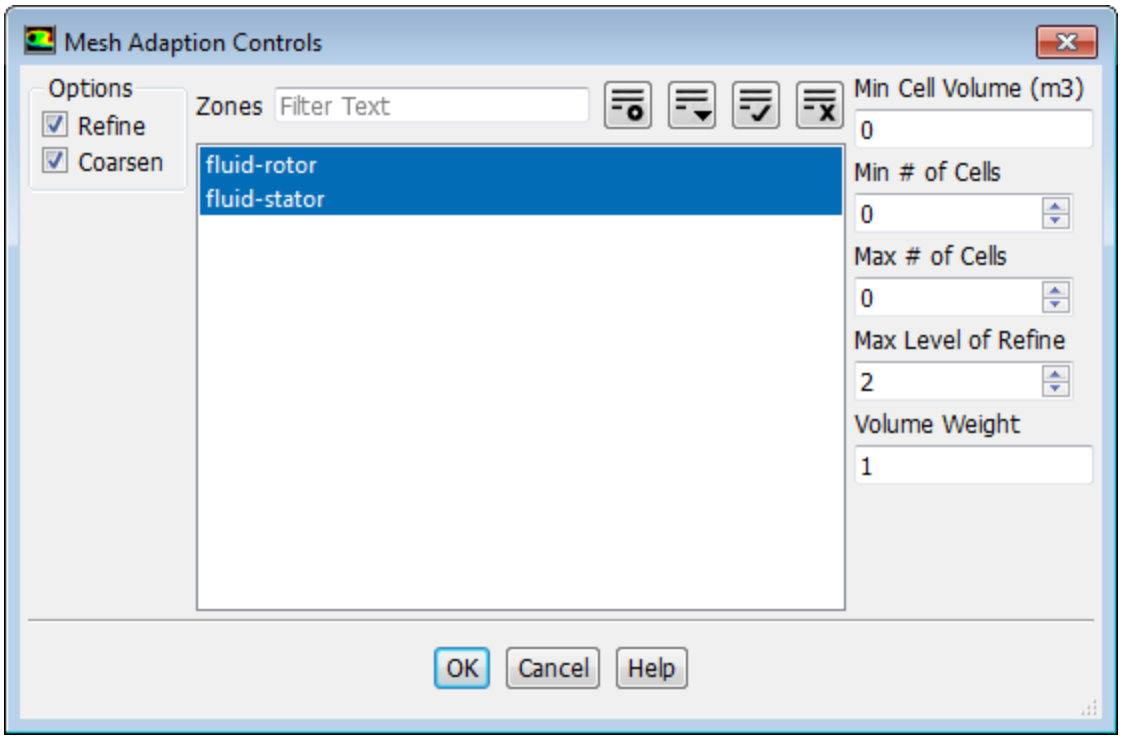
Merge
merges the selected zones into a single zone. If your case file has dynamic zones or mesh interfaces, the Warning Dialog Box (p. 2841) will open and require your input prior to the merge.

38.2.50. Mesh Adaption Controls Dialog Box

The **Mesh Adaption Controls** dialog box allows you to set limits on the minimum cell size, the minimum and maximum number of cells, and the cell types that can be adapted. In addition, it allows you to vary the volume weighting of the gradient adaption function. See Mesh Adaption Controls (p. 1705) for details about the items below.

Setting Up Domain → Adapt → More → Controls...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options
contains check buttons that control the manner in which the mesh can be adapted.

Refine
toggles mesh adaption by adding points.

Coarsen
toggles mesh adaption by removing points.

Zones
contains a list of cell zones from which you can select the zones in which to perform adaption (or marking). By default, all cell zones are selected.

Min Cell Volume
restricts the size of the cell that is considered for refinement. Even if the cell is marked for refinement, it will not be refined if its cell volume is less than this threshold value.

Min # of Cells
specifies the minimum number of cells required in the mesh.

Note

When using the parallel solver, the **Min # of Cells** value is not strictly obeyed, but provides an approximate limit to the minimum cell count that the adaption algorithm will allow.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Max # of Cells
limits the total number of cells allowed in the mesh. A value of zero places no limits on the number of cells.

> **Note**
>
> When using the parallel solver, the **Max # of Cells** value is not strictly obeyed, but provides an approximate limit to the maximum cell count that the adaption algorithm will allow.

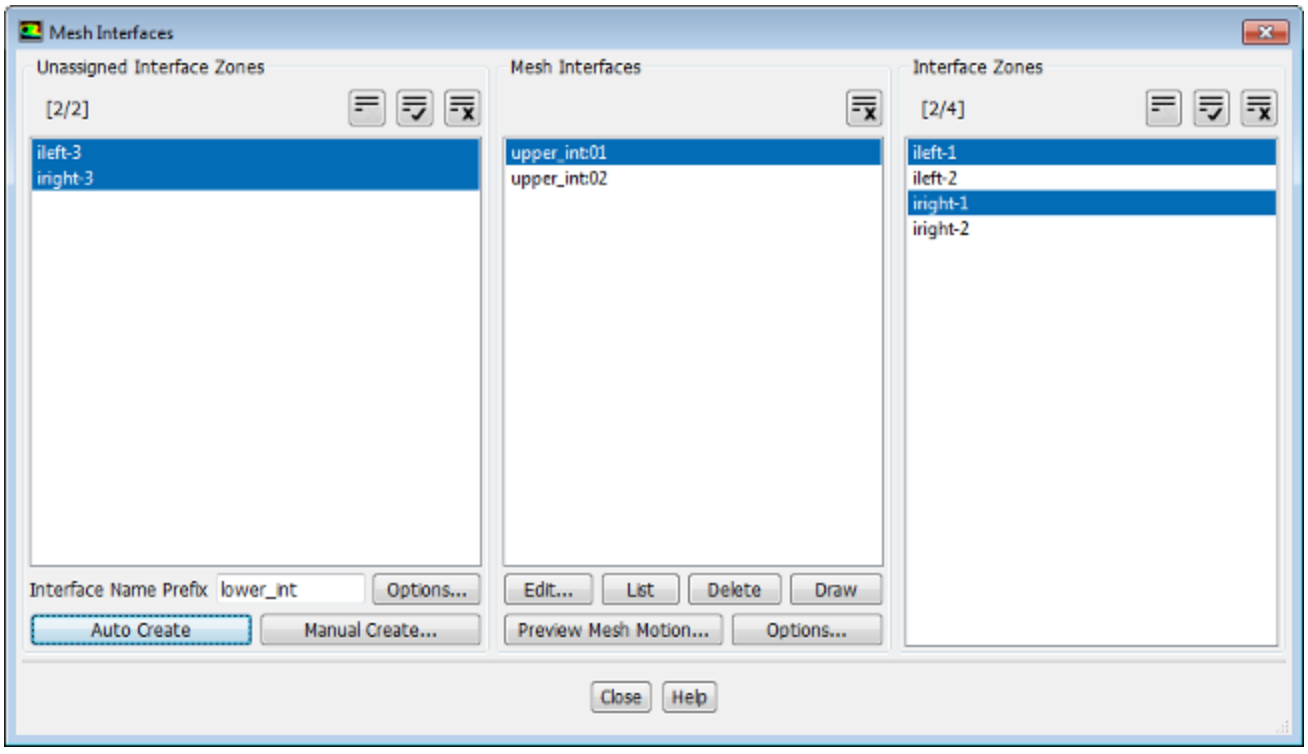
Max Level of Refine
specifies the maximum level of refinement for the cells.

Volume Weight
controls the volume weighting in the gradient adaption function. Valid values are between 0 and 1: 0 for no volume weighting and 1 for full volume weighting.

38.2.51. Mesh Interfaces Dialog Box

The **Mesh Interfaces** dialog box allows you to automatically or manually create mesh interfaces for use with sliding meshes (see Using Sliding Meshes (p. 633)) or multiple reference frames (see Mesh Setup for a Multiple Moving Reference Frame (p. 616)), or for meshes with non-conformal boundaries (see Non-Conformal Meshes (p. 150)). For information about using this dialog box, see Using a Non-Conformal Mesh in ANSYS Fluent (p. 164).

Setting Up Domain → Interfaces → Mesh...



Controls

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Unassigned Interface Zones

allows you to create new mesh interfaces using the automatic or manual method. Note that the selection list at the top of the group box (which lists interface zones that have not yet been assigned to an existing mesh interface) is only used as part of the automatic method.

Interface Name Prefix

specifies the prefix used in the names of all the mesh interfaces created automatically through the **Auto Create** button.

Options...

opens the Auto Create Options Dialog Box (p. 2672), so you can define settings that you want applied to the mesh interfaces that are created automatically through the **Auto Create** button.

Auto Create

creates mesh interfaces using the automatic method, so that Fluent is responsible for determining which of the selected **Unassigned Interface Zones** can be grouped together to form the two sides of one or more mesh interfaces. Note that mesh interfaces that use the periodic or periodic repeat option cannot be created using this button, but must instead be created through the **Manual Create...** button.

Manual Create...

opens the Create/Edit Mesh Interfaces Dialog Box (p. 2680), so you can create mesh interfaces using the manual method. Note that this method is only necessary for mesh interfaces that use the periodic or periodic repeat option.

Mesh Interfaces

allows you to manage existing mesh interfaces. Note that many of the buttons in this group box are not accessible unless you make a selection from the list at the top.

Edit...

opens the Edit Mesh Interfaces Dialog Box (p. 2696), so you can edit the name, the list of interface zones assigned to the interface, and/or the interface options.

List

prints information about the selected mesh interface in the console. When you click this button, ANSYS Fluent will list the two interface boundaries and (if you have initialized the solution) all new zones that were created (that is, wall and/or interior zones).

Delete

deletes the selected mesh interface. Note that this button can only be used on one mesh interface at a time; if you would instead like to delete multiple interfaces in a single action, you can select them in the tree and use the right-click menu command.

Draw

allows you to display the selected mesh interface (along with any additional selections in the **Unassigned Interface Zones** and/or **Interface Zones** lists) in the graphics window.

Preview Mesh Motion...

opens the Mesh Motion Dialog Box (p. 2500).

Options...

opens the Interface Options Dialog Box (p. 2717), where you can set global parameters for mapped mesh interfaces.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Interface Zones

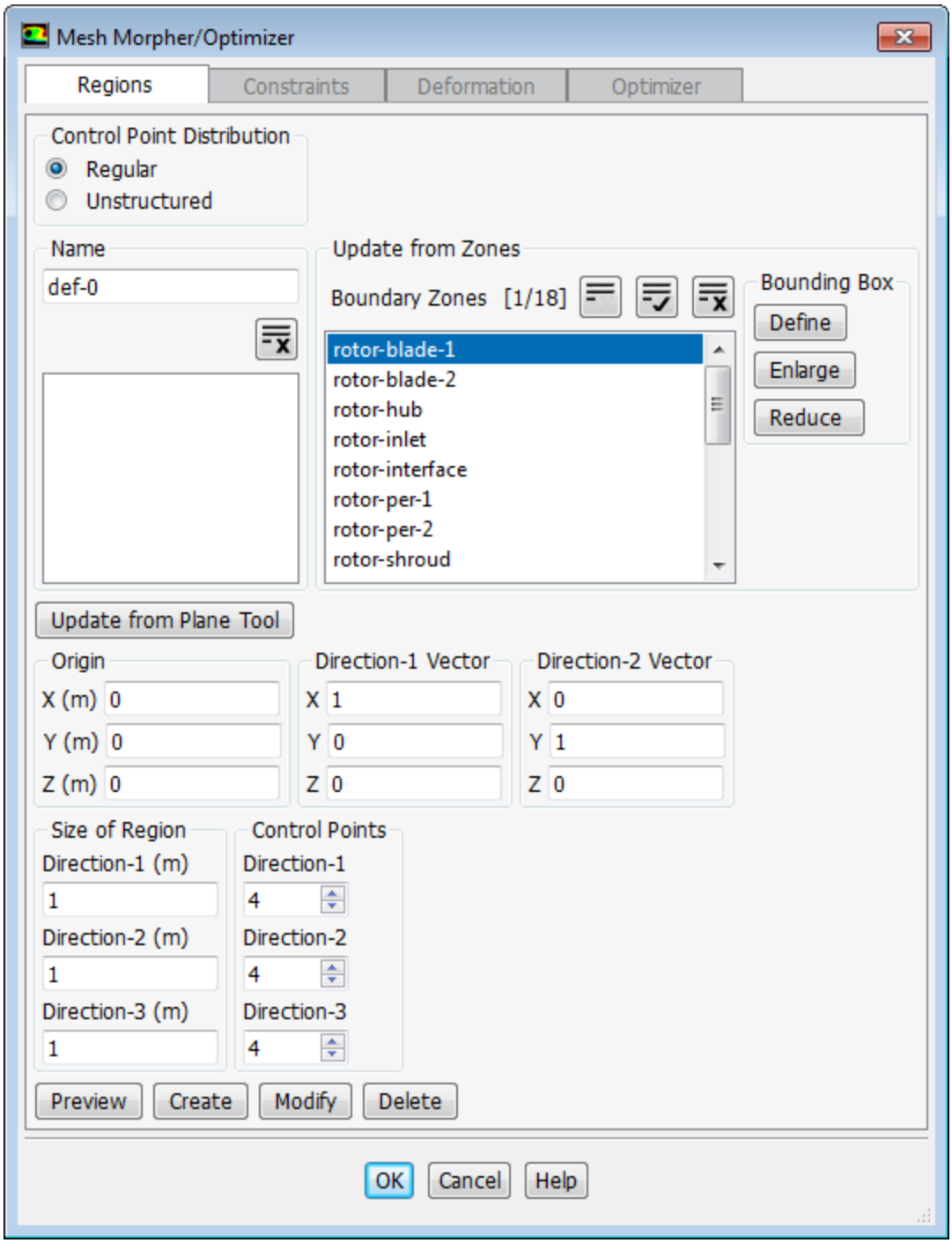
highlights the interface zones assigned to the mesh interface selected in the **Mesh Interfaces** group box, and allows you to select specific interface zones to be displayed in the graphics window using the **Draw** button.

38.2.52. Mesh Morpher/Optimizer Dialog Box

The **Mesh Morpher/Optimizer** dialog box allows you to use the mesh morpher/optimizer to deform the mesh in order to solve shape optimization problems. You can deform the mesh manually, allow a built-in optimizer to define the deformation in order to minimize an objective function, or use Design Exploration in ANSYS Workbench to easily explore a variety of deformation scenarios. See Using the Mesh Morpher/Optimizer (p. 2125) for further details.

Design → **Parameter-Based** → **Optimizer...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Regions

tab allows you to define the regions of the domain where the mesh will be deformed in order to optimize the shape.

Control Point Distribution

allows you to select the way in which you will define the locations of the control points, as well as the kinds of motions you can apply.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Regular
specifies that the control points will be spread in a regular distribution throughout the entire deformation region. The number of control points along each of the defining direction vectors will be specified in the **Control Points** group box. The motion of the control points will be limited to translations, as the ability to define rotational or radial motions will not be available.

Unstructured

specifies that control point locations will be specified by right-clicking with the mouse on boundary zones, distributing an approximate number throughout a zone, or by entering coordinates, using the Define Control Points Dialog Box (p. 2685). The motion of the control points will be able to be defined as translations, as rotations about a point / axis, or radially about a point / axis.

Name
provides controls for the deformation regions you are defining. The upper text-entry box defines the name of the region that is created when you click the **Create** button. The lower selection list allows you to select a region that has already been created, in order to display it in the graphics window, modify the settings via the **Modify** button, or delete the region via the **Delete** button.

Update from Zones
allows you to create a bounding box that defines the deformation region.

Boundary Zones
specifies the boundary zones that will be used to generate a bounding box for the deformation region when you click the **Define** button.

Define
updates the values in the **Origin**, **Direction-1 Vector**, **Direction-2 Vector**, and **Size of Region** group boxes based on a bounding box that encompasses the selections in the **Boundary Zones** list, and previews the bounding box in green in the graphics window.

Enlarge
increases the size of the bounding box created by the **Define** button, and previews the revised bounding box in green in the graphics window.

Reduce
decreases the size of the bounding box created by the **Define** button, and previews the revised bounding box in green in the graphics window.

Update from Plane Tool
updates the values in the **Direction-1 Vector** and **Direction-2 Vector** group boxes based on the settings in the plane tool. See Using the Plane Tool (p. 1724) for information about the plane tool. This button is only available for 3D cases.

Update from Line Tool
updates the values in the **Direction-1 Vector** and **Direction-2 Vector** group boxes based on the settings in the line tool. See Using the Line Tool (p. 1720) for information about the line tool. This button is only available for 2D cases.

Origin
allows you to define the Cartesian coordinates of the origin of the deformation region.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X

defines the *x* coordinate of the origin of the deformation region.

Y

defines the *y* coordinate of the origin of the deformation region.

Z

defines the *z* coordinate of the origin of the deformation region. This number-entry box is only available for 3D cases.

Direction-1 Vector

allows you to define a vector that acts as the first direction of the deformation region relative to the **Origin**.

X

defines the *x* component of the first direction vector of the deformation region.

Y

defines the *y* component of the first direction vector of the deformation region.

Z

defines the *z* component of the first direction vector of the deformation region. This number-entry box is only available for 3D cases.

Direction-2 Vector

defines a vector that is used to calculate the second direction of the deformation region relative to the **Origin**. For 2D cases, the values in the **Direction-2 Vector** group box are automatically defined by ANSYS Fluent to be perpendicular to the **Direction-1 Vector**. For 3D cases, the vector defined by the **Direction-2 Vector** group box is projected onto a plane that intersects the **Origin** and is perpendicular to the vector defined in the **Direction-1 Vector** group box, in order to calculate the second direction.

X

defines the *x* component of the vector used to calculate the second direction of the deformation region. This number-entry box is only editable for 3D cases.

Y

defines the *y* component of the vector used to calculate the second direction of the deformation region. This number-entry box is only editable for 3D cases.

Z

defines the *z* component of the vector used to calculate the second direction of the deformation region. This number-entry box is only available for 3D cases.

Size of Region

allows you to define the overall dimensions of the deformation region.

Direction-1

defines the length of the deformation region along the direction specified by the **Direction-1 Vector** group box.

Direction-2

defines the length of the deformation region along the direction specified by the **Direction-2 Vector** group box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Direction-3
defines the length of the deformation region along the third axis (that is, the cross product of the vectors defined by the **Direction-1 Vector** and **Direction-2 Vector** group boxes). This number-entry box is only available for 3D cases.

Boundary Continuity
allows you to specify whether the mesh transitions are smooth or abrupt wherever the mesh intersects with the limits of the deformation region. This group box is only available when **Unstructured** is selected from the **Control Point Distribution** list.

Smooth Transitions
enables smooth mesh transitions wherever the mesh intersects with the limits of the deformation region, based on the creation of stationary control points at those intersections.

Display Control Points
displays the stationary control points created when the **Smooth Transitions** option is enabled.

Control Points
allows you to define the control points for the deformation regions.

Direction-1
defines the number of control points for the deformation region selected in the **Name** group box along the direction specified by the **Direction-1 Vector** group box. This number-entry box is only available when **Regular** is selected from the **Control Point Distribution** list.

Direction-2
defines the number of control points for the deformation region selected in the **Name** group box along the direction specified by the **Direction-2 Vector** group box. This number-entry box is only available when **Regular** is selected from the **Control Point Distribution** list.

Direction-3
defines the number of control points for the deformation region selected in the **Name** group box along the third axis (that is, the cross product of the vectors defined by the **Direction-1 Vector** and **Direction-2 Vector** group boxes). This number-entry box is only available for 3D cases when **Regular** is selected from the **Control Point Distribution** list.

Define...
opens the Define Control Points Dialog Box (p. 2685), which you can use to define the control points. This button is only available when **Unstructured** is selected from the **Control Point Distribution** list.

Preview
displays (in green) the bounding box that results from the current settings in the **Origin**, **Direction-1 Vector**, **Direction-2 Vector**, and **Size of Region** group boxes.

Create
creates a new deformation region with the name specified in the upper text-entry box of the **Name** group box and with the settings defined using the **Origin**, **Direction-1 Vector**, **Direction-2 Vector**, **Size of Region**, and (for regular control point distributions) **Control Points** group boxes. The newly created deformation region will be added to and selected in the lower **Name** selection list, and will be displayed (in blue) in the graphics window.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modify
modifies the saved settings for the deformation region selected in the lower **Name** selection list and displays the modified region in the graphics window.

Delete
deletes the deformation region selected in the lower **Name** selection list.

Constraints
tab allows you to define the constraints on the boundary zones, in order to limit the freedom of particular zones that fall within the deformation region(s) during the morphing of the mesh.

Zones
allows you to select the boundary zones for which you are defining a constraint.

Option
specifies the constraint on the boundary zones selected from the **Zones** list.

Unconstrained
specifies that the boundary zones are completely free to be deformed according to the assigned parameters.

Passive
specifies that the nodes of the boundary zones are partially constrained to varying degrees, based on their proximity to adjacent boundary zones that are fixed. The nodes in a passive boundary zone behave in a similar manner to the interior mesh nodes, in order to ensure that there is a smooth transition between fixed and unconstrained boundary zones.

Fixed
specifies that the boundary zones are fixed and will not be deformed.

Display
displays the selected **Zones** in the graphics window.

Summary
prints a list in the console that summarizes the constraint definitions for all of the boundary zones.

Deformation
tab allows you to define the deformation.

Number of Parameters
defines the number of parameters available to define the motion of the control points.

Parameter Values
allows you to define either the magnitude of deformation associated with each parameter (when you are manually specifying the deformation or using Design Exploration in ANSYS Workbench to explore multiple deformation scenarios) or the bounds for each parameter (when you are using a built-in optimizer).

par1, par2...
specify the magnitude of deformation associated with each parameter. You can enter a numeric value in the field or use the icon to create or assign an input parameter (see Select Input Parameter Dialog Box (p. 2384) for details). These fields and icons are only available when **none** or **workbench** is selected from the **Optimizer** drop-down list in the **Optimizer** tab. Note that the value entered here will specify a length in meters for translations and radial motions, and will

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

specify an angle in degrees for rotations; these units will be used regardless of what units you are using in the case.

Apply
saves definitions for the parameter fields in the **Parameter Values** group box. This button is only available when **none** or **workbench** is selected from the **Optimizer** drop-down list in the **Optimizer** tab.

Set Bounds...
opens the Parameter Bounds Dialog Box (p. 2760), where you can define strict minimum and maximum values for the parameters. Note that this button is not available when **workbench** is selected from the **Optimizer** drop-down list in the **Optimizer** tab.

Motion Settings...
opens the Motion Settings Dialog Box (p. 2749), where you can define the motion of the control points.

Deform
applies the current settings in the **Parameter Values** group box, modifies the mesh, and updates the mesh display in the graphics window. This button is only available when **none** is selected from the **Optimizer** drop-down list in the **Optimizer** tab.

Check
displays a mesh check report in the console for the mesh displayed in the graphics window. This button is only available when **none** is selected from the **Optimizer** drop-down list in the **Optimizer** tab. The mesh check report provides volume statistics, mesh topology and periodic boundary information, verification of simplex counters, and verification of node position with reference to the x axis for axisymmetric cases, in the same manner as the **Check** button in the **General** task page (see Checking the Mesh (p. 192) for details).

Reset
undoes any modifications made to the mesh as a result of the **Deform** button and updates the mesh display in the graphics window.

Optimizer
tab allows you to define the settings for the optimizer.

Optimizer
specifies which optimizer is used. The choices include **none** (for manual deformation); **compass**, **newuoa**, **powell**, **rosenbrock**, **simplex**, and **torczon** (for optimization in ANSYS Fluent); and **workbench** (for optimizing using Design Exploration in ANSYS Workbench). Note that **workbench** is only available if you have launched your ANSYS Fluent session from ANSYS Workbench. For information about how the built-in optimizers function or how to use Design Exploration, see Design Analysis and Optimization in the User's Guide or Working With Input and Output Parameters in Workbench in the ANSYS Fluent in Workbench User's Guide, respectively.

Objective Function Definition...
opens the Objective Function Definition Dialog Box (p. 2757), where you can specify the format of the objective function that will be minimized during the optimization process. This button is not available if **none** or **workbench** is selected from the **Optimizer** drop-down list.

Optimizer Settings
allows you to specify the optimizer settings. This group box is not available if **none** or **workbench** is selected from the **Optimizer** drop-down list.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Maximum Number of Designs
defines the maximum number of design stages the optimizer will undergo to reach the specified objective function.

Maximum Iterations per Design
defines the maximum number of iterations ANSYS Fluent will perform for each design change.

Optimizer Convergence Criteria
defines the convergence criteria for the optimizer.

Initial Parameter Variation
defines how much the parameters will be allowed to vary during the initial calculations when **newuoa** is selected from the **Optimizer** drop-down list.

Mesh Quality
allows you to enable and define a mesh quality check: if the orthogonal quality (as defined in Mesh Quality (p. 131)) for any cell of a mesh is less than a specified value, no solution is calculated for the mesh and Fluent proceeds to the next design stage. This group box is not available if **none** is selected from the **Optimizer** drop-down list.

Reject Poor Quality Meshes
enables the consideration of orthogonal quality when determining whether a solution should be calculated for a mesh.

Minimum Orthogonal Quality
defines the minimum orthogonal quality value every cell must have in order for a solution to be calculated for a mesh. Values may range from `0`–`1` (where `0` represents the worst quality).

Case and Data File Sets
allows you to enable the automatic saving of intermediate case and data files at specified intervals during the optimization run, so that you can restart an interrupted solution in the same or a different Fluent session without increasing the overall number of design iterations needed to reach convergence. This group box is not available if **none** or **workbench** is selected from the **Optimizer** drop-down list.

Save Every
specifies the frequency (in number of design iterations) that you want intermediate case and data file sets saved. Note that the default value is **0**, which specifies that no intermediate files will be saved.

Maximum Number Retained
specifies the maximum number of intermediate case and data files sets you want to retain. After the maximum limit of file sets has been saved, ANSYS Fluent begins overwriting the earliest existing intermediate file set.

File Name
specifies a root name for the intermediate case and data file sets, which will be appended with a number that represents the design iteration during which it was saved. Note that you can include a folder path in the file name if you want the files saved outside of the working folder.

Initialization
provides settings that determine how the solution variables are treated after the mesh is deformed during the optimization process. This group box is not available if **none** or **workbench** is selected from the **Optimizer** drop-down list.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Method
allows you to select one of the following:

Initialize Data After Morphing
specifies that the solution variables should be initialized to the values defined in the **Solution Initialization** task page after deformation.

Continue with Current Data
specifies that the solution variables should remain the values obtained in the previous design iteration.

Read Data File After Morphing
specifies that the solution variables are set to the values obtained from the data file specified in the **Data File Name** text-entry box after deformation. This selection is not available when **newuoa** is selected from the **Optimizer** drop-down list.

Data File Name
specifies the data file from which the solution variables are obtained when **Read Data File After Morphing** is selected.

Execute Commands
allows you to specify commands (text commands or command macros) that will be run during the optimization runs of the mesh morpher/optimizer. This group box is not available if **none**, **newuoa**, or **workbench** is selected from the **Optimizer** drop-down list.

Initial Commands
specifies the commands that will be run after the design has been modified, but before ANSYS Fluent has started to run the calculation for that design stage.

End Commands
specifies the commands that will be run after the solution has run and converged for a design stage.

Monitor...
opens the Optimization History Monitor Dialog Box (p. 2759), which allows you to plot and/or record how the value of the objective function varies with each design stage. This button is not available if **none**, **newuoa**, or **workbench** is selected from the **Optimizer** drop-down list.

Apply
saves the settings in the **Optimizer** tab.

Summary
displays a summary of the mesh morpher/optimizer settings in the console.

Optimize
initiates the optimization process using the settings saved in all of the tabs of the **Mesh Morpher/Optimizer** dialog box. This button is not available if **none** or **workbench** is selected from the **Optimizer** drop-down list.

38.2.53. MiniVAS Settings Dialog Box

The **MiniVAS Settings** dialog box allows you to set specific MiniVAS options.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **MiniVAS Settings** dialog box is opened by clicking the **Settings...** button in the Video Control Dialog Box (p. 2833), after you have selected MiniVAS as your video device protocol.

Controls

VTR Identification

identifies the type of VTR the MiniVAS is connected to. The identification is a two-character code reserved for your VTR model (see your MiniVAS documentation for details). If left blank, this field is ignored and the MiniVAS will use the code that has been permanently stored in its ROM. It is recommended that you leave this field blank unless you know that a code change is required. If you open this dialog box after opening a connection to the MiniVAS, this field will contain the code that is stored in the MiniVAS.

VTR Video Format

identifies the video format standard to which your VTR conforms. This setting is used to determine the video frame rate (frames/sec).

NTSC

(National Television Standards Committee) is the main video format used in North America and Japan. The video frame rate is 30 frames/sec.

PAL

(Phase Alternate Line) is the video format used by some countries in Western Europe, Asia, and Australia. The video frame rate is 25 frames/sec.

VTR Tape Format

specifies which counter format to use for editing. MiniVAS provides commands based on both time code and frame code.

Time Code

instructs MiniVAS to control the VTR using time code.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Frame Code
instructs MiniVAS to control the VTR using frame code.

Preroll
specifies the preroll time in seconds. This is used when performing **Edit** records, including frame-by-frame animation recording, to make sure the tape is moving at the proper speed through the recorded segment. The value given is the number of seconds the tape will play up to the record in point. The default setting should be sufficient for most VTR devices.

Preblack Mode
specifies the mode used when you perform a preblack recording session.

Preblack
instructs the MiniVAS to perform a "hard" record on a previously unformatted tape. During this operation, the MiniVAS will command the VTR to write both a time code and a VIFC frame code to the tape.

Prestripe
instructs the MiniVAS to perform an "edit" record on a previously formatted tape that already has a time code. During this operation, the MiniVAS will command the VTR to write a VIFC frame code to the tape.

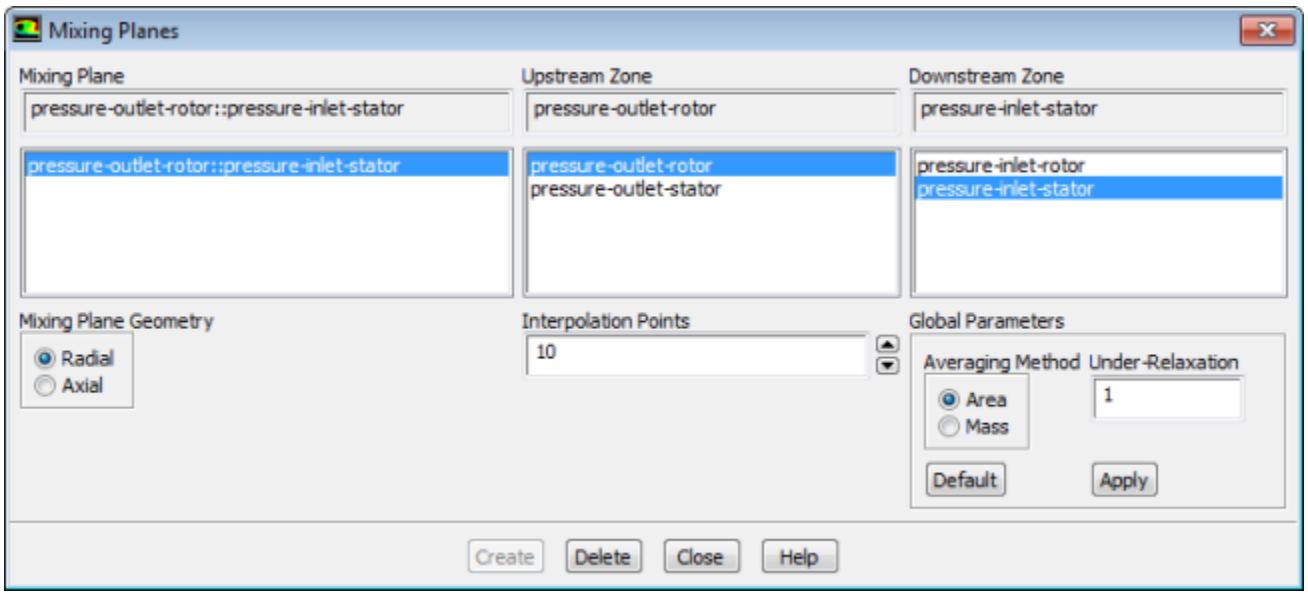
In either case, any previously recorded video will be destroyed.

Verbose Mode
enables the echoing of each MiniVAS command as it is sent to the MiniVAS controller.

38.2.54. Mixing Planes Dialog Box

The **Mixing Planes** dialog box allows you to define the mixing planes for a mixing plane model. See Setting Up the Mixing Plane Model (p. 619) for details about using the items below.

Setting Up Domain → Mesh Models → Mixing Planes...



Controls

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mixing Plane
contains a list from which you can select an existing mixing plane, and an informational field in which ANSYS Fluent displays the name of the currently selected (or most recently created) mixing plane.

Upstream Zone, Downstream Zone
contain lists from which you can select the boundaries on the upstream and downstream sides of the mixing plane, and informational fields that show the names of the zone you selected in each list. (You cannot edit these fields; the name in each field will be the name of the zone you selected in the list below it.)

Interpolation Points
specifies the number of radial or axial locations used in constructing the boundary profiles for circumferential averaging. This item appears only in 3D.

Mixing Plane Geometry
defines the geometry of the mixing plane interface. This item appears only in 3D.

Radial
specifies that information at the mixing plane interface is to be circumferentially averaged into profiles that vary in the radial direction, for example, $p(r)$, $T(r)$.

Axial
specifies that circumferentially averaged profiles are to be constructed that vary in the axial direction, for example, $p(x)$, $T(x)$.

Global Parameters
contains parameters related to the mixing plane calculation.

Averaging Method
consists of three profile averaging methods.

Area
is the default method and is expressed using Equation 2.17 in the Theory Guide.

Mass
provides better representation of the total quantities than the area averaging method. It is defined by Equation 2.18 of the Theory Guide.

Mixed-Out
is most representative of non-uniform flow profiles and is expressed using Equation 2.20 in the Theory Guide.

Under-Relaxation
specifies the under-relaxation factor for updating the boundary values at mixing planes.

Apply
sets the specified **Under-Relaxation**.

Default
sets the **Under-Relaxation** to its default value, as assigned by ANSYS Fluent. After execution, the **Default** button becomes the **Reset** button.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reset
resets the **Under-Relaxation** to its most recently saved value (that is, the value before **Default** was selected). After execution, the **Reset** button becomes the **Default** button.

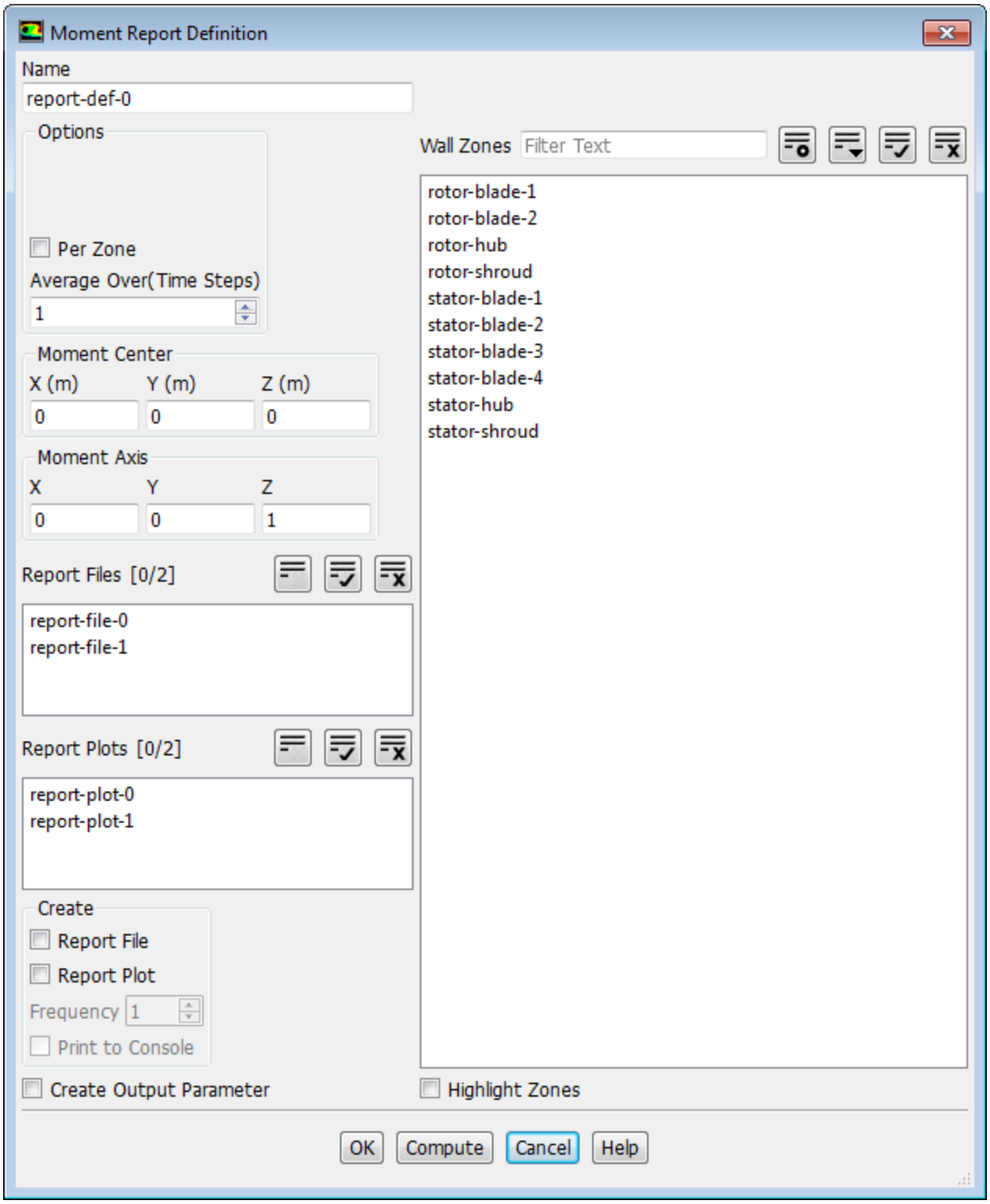
Create
creates the specified mixing plane (and assigns it a name in the **Mixing Plane** field).

Delete
deletes the mixing plane selected under **Mixing Plane**.

38.2.55. Moment Report Definition Dialog Box

Solving → **Reports** → **Definitions** → **New** → **Force Report** → **Moment...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options

Per Zone

specifies whether or not moments for multiple walls should be monitored on each wall separately.When this option is turned off (the default), Fluent computes and monitors the total force for all of the selected walls combined together.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Average Over (Time-Steps), Average Over (Iterations)
(optional) To have Fluent calculate a running average for the **Moment Report Definition** you can enter a positive integer greater than 1 (the default) for **Average Over**.

Specifying a number greater than 1 means that Fluent will print, plot, and write the running average value of the selected variable instead of the current value of the same variable.

The value reported is averaged over the last N *iterations/time steps, where* N *is your specified* ***Average Over*** *value. When the iteration/time step number is lower than* N*, Fluent calculates the average of the available variable values.*

Moment Center
contains the Cartesian coordinates of the moment center.

Moment Axis
contains the Cartesian coordinates of the moment vector to be monitored.

Report Files
lists all of the report files where you can write the report definition data.

Report Plots
lists all of the report plots where you can plot the report definition data (assuming this report has the same units as the other report definitions included in the selected report plot).

Create

Report File
creates a new report file that includes this report definition.

Report Plot
creates a new report plot that includes this report definition.

Frequency
specifies how frequently the report definition is written, plotted, or printed to the console.

Print to Console
prints the value of the report definition to the console.

Create Output Parameter
creates an output parameter for this report definition with the name `<report definition name>-op`.

Wall Zones
lists the zones you can select for moment computation.

Highlight Zones
when enabled, highlights the selected **Wall Zones** in the graphics window.

OK
creates the report definition.

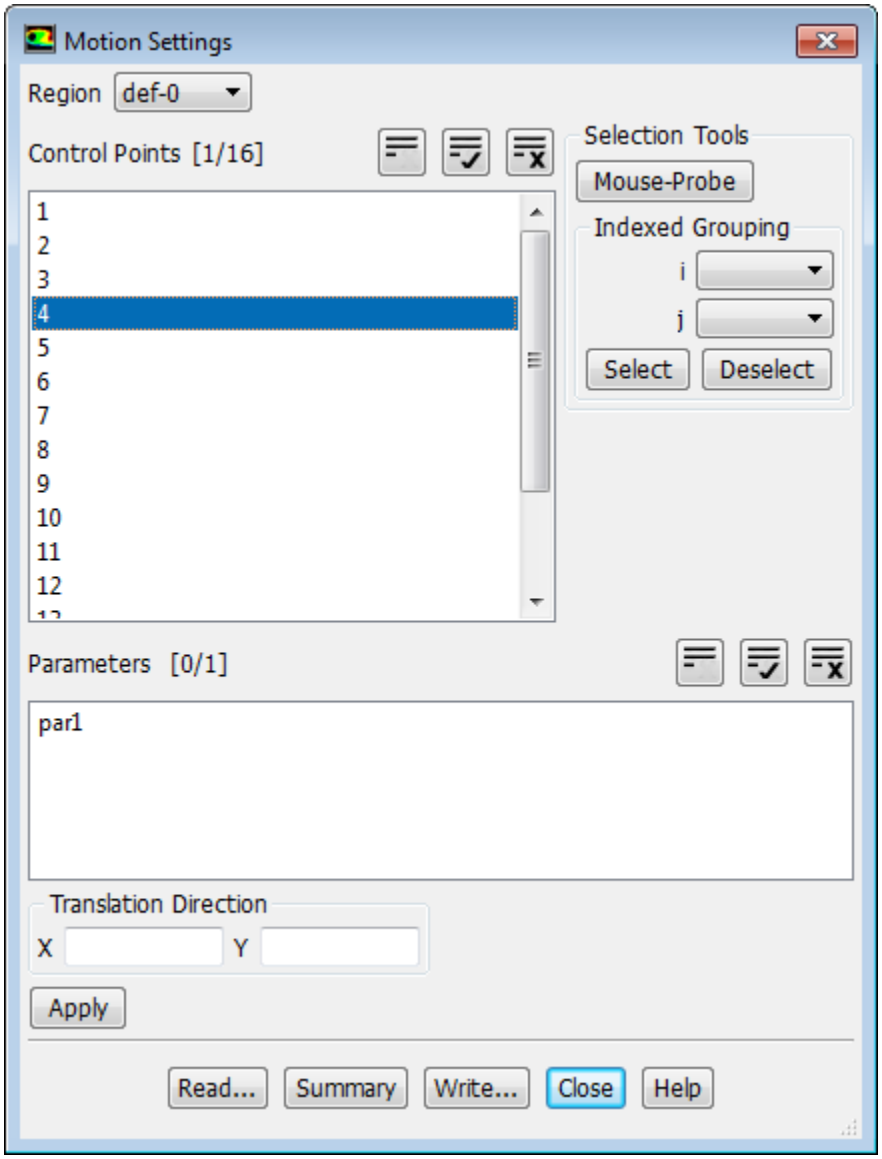
Compute
computes the value of the report definition at the selected zone(s) and prints to the console.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.2.56. Motion Settings Dialog Box

The **Motion Settings** dialog box allows you to define the motion of the control points, as part of the definition of the deformation produced by the mesh morpher/optimizer. Note that the controls and workflow of this dialog box are very much dependent on the selection in the **Control Point Distribution** list in the **Regions** tab of the **Mesh Morpher/Optimizer** dialog box. See Using the Mesh Morpher/Optimizer (p. 2125) for details about using this dialog box.

The **Motion Settings** dialog box is opened from the Mesh Morpher/Optimizer Dialog Box (p. 2734).



Controls (when **Regular** is selected from the **Control Point Distribution** list in the **Regions** tab of the **Mesh Morpher/Optimizer** dialog box)

Region

selects the deformation region for which you are assigning parameters and translation directions to control points.

Control Points

defines the control points to which you are assigning parameters and translation directions.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selection Tools

provides tools for selecting the control points to which you are assigning parameters and translation directions.

Mouse-Probe

allows you to specify the **Control Points** by clicking in the graphics window with the mouse-probe button (which is the right mouse button, by default).

Indexed Grouping

allows you to select the control points based on their assigned indices.

i

allows you to select from a list of control point index numbers, which identify the location in a sequence (starting at the origin of the region) of control points along the direction-1 vector (as defined in the **Regions** tab of the **Mesh Morpher/Optimizer** dialog box).

j

allows you to select from a list of control point index numbers, which identify the location in a sequence (starting at the origin of the region) of control points along the direction-2 vector (as defined in the **Regions** tab of the **Mesh Morpher/Optimizer** dialog box).

k

allows you to select from a list of control point index numbers, which identify the location in a sequence (starting at the origin of the region) of control points along the direction-3 vector (as defined—though not displayed—in the **Regions** tab of the **Mesh Morpher/Optimizer** dialog box).

Select

adds the control points that satisfy the selected index criteria (that is, the selections from the **i**, **j**, and **k** drop-down lists) to the list of selected **Control Points**.

Deselect

removes the control points that satisfy the selected index criteria (that is, the selections from the **i**, **j**, and **k** drop-down lists) from the list of selected **Control Points**.

Parameters

specifies the names of the parameters (whose values are displayed in the **Parameter Values** group box in the **Deformation** tab of the **Mesh Morpher/Optimizer** dialog box) that you are assigning to the **Control Points**. The products of the values associated with these parameters and the settings specified in the **Translation Direction** group box define the displacement of the control points during manual deformation.

Translation Direction

allows you to define coefficients in the *x*, *y*, and (for 3D cases) *z* directions to scale the magnitude of deformation for the selected parameters. If you use a built-in optimizer, these coefficients provide the direction of the displacement of the control points and the optimizer determines the overall magnitude of displacement. Alternatively, if you manually specify the deformation or use Design Exploration in ANSYS Workbench to explore multiple deformation scenarios, the values you enter in the **Translation Direction** group box are multiplied with the values of the **Parameters** (as displayed in the **Parameter Values** group box in the **Deformation** tab of the **Mesh Morpher/Optimizer** dialog box) to define the displacement of the control points.

X

defines the scaling coefficient applied to the deformation parameter in the *x* direction.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Y
defines the scaling coefficient applied to the deformation parameter in the *y* direction.

Z
defines the scaling coefficient applied to the deformation parameter in the *z* direction. This number-entry box is only available for 3D cases.

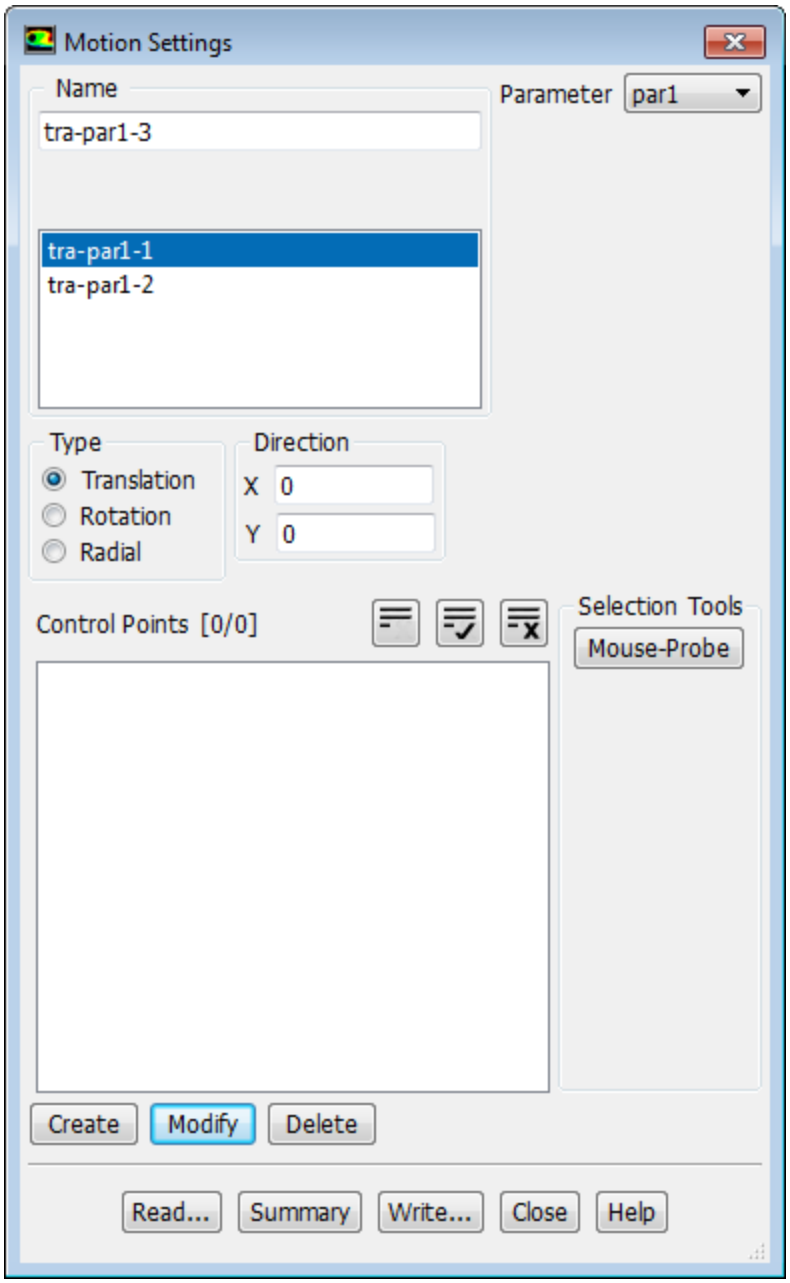
Apply
saves the selections made in the **Parameters** selection list and the values entered in the **Translation Direction** group box, and assigns these parameter settings to the control points selected in the **Control Points** selection list.

Read...
allows you to define your motion settings by reading an ASCII text file. See Mesh Morpher/Optimizer File Formats (p. 2867) for further details about the text file format.

Summary
displays a summary of the saved motion settings in the console.

Write...
allows you to write your saved motion settings to an ASCII text file. See Mesh Morpher/Optimizer File Formats (p. 2867) for further details about the text file format.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls (when **Unstructured** is selected from the **Control Point Distribution** list in the **Regions** tab of the **Mesh Morpher/Optimizer** dialog box)

Name

provides controls for the motions you are defining. The upper text-entry box defines the name of the motion that is created when you click the **Create** button (which by default is based on the type of motion and the related parameter). The lower selection list allows you to select a motion that has already been created, in order to display the selections associated with it, or to modify or delete the motion using the **Modify** or **Delete** buttons, respectively.

Parameter

allows you to select the parameter (whose value is displayed in the **Parameter Values** group box in the **Deformation** tab of the **Mesh Morpher/Optimizer** dialog box) for the motion.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type
allows you to specify the type of motion you want to define.

Translation
specifies that the motion is a translation, based on the settings in the **Direction** group box.

Rotation
specifies that the motion is a rotation about a point (for 2D) or an axis (for 3D), based on the settings in the **Axis Origin** group box and (for 3D cases) **Axis Direction** group box.

Radial
specifies that the motion is defined radially about a point (for 2D) or an axis (for 3D), based on the settings in the **Axis Origin** group box and (for 3D cases) **Axis Direction** group box.

Direction
allows you to define coefficients in the **X**, **Y**, and (for 3D cases) **Z** directions to scale the magnitude of deformation for the selected parameter. If you use a built-in optimizer, these coefficients provide the direction of the displacement of the control points and the optimizer determines the overall magnitude of displacement. Alternatively, if you manually specify the deformation or use Design Exploration in ANSYS Workbench to explore multiple deformation scenarios, the values you enter in the **Direction** group box are multiplied with the values of the **Parameter** (as displayed in the **Parameter Values** group box in the **Deformation** tab of the **Mesh Morpher/Optimizer** dialog box) to define the displacement of the control points.

This group box is only available when **Translation** is selected from the **Type** list.

Axis Direction
allows you to define the **X**, **Y**, and **Z** components of the vector about which the control points will rotate or move radially, when the selection in the **Type** list is **Rotation** or **Radial**, respectively. This group box is only available for 3D cases.

Axis Origin
allows you to define the **X**, **Y**, and (for 3D cases) **Z** coordinates of the origin about which the control points will rotate or move radially, when the selection in the **Type** list is **Rotation** or **Radial**, respectively.

Control Points
allows you to select the control points that will undergo the motion.

Selection Tools
provides tools for selecting the control points that will undergo the motion.

Mouse-Probe
allows you to specify the **Control Points** by clicking in the graphics window with the mouse-probe button (which is the right mouse button, by default).

Create
creates a new motion based on the currently displayed settings.

Modify
modifies the saved settings for the motion selected in the lower **Name** selection list.

Delete
deletes the motion selected in the lower **Name** selection list.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Read...
allows you to define your motion definitions by reading an ASCII text file. See Mesh Morpher/Optimizer File Formats (p. 2867) for further details about the text file format.

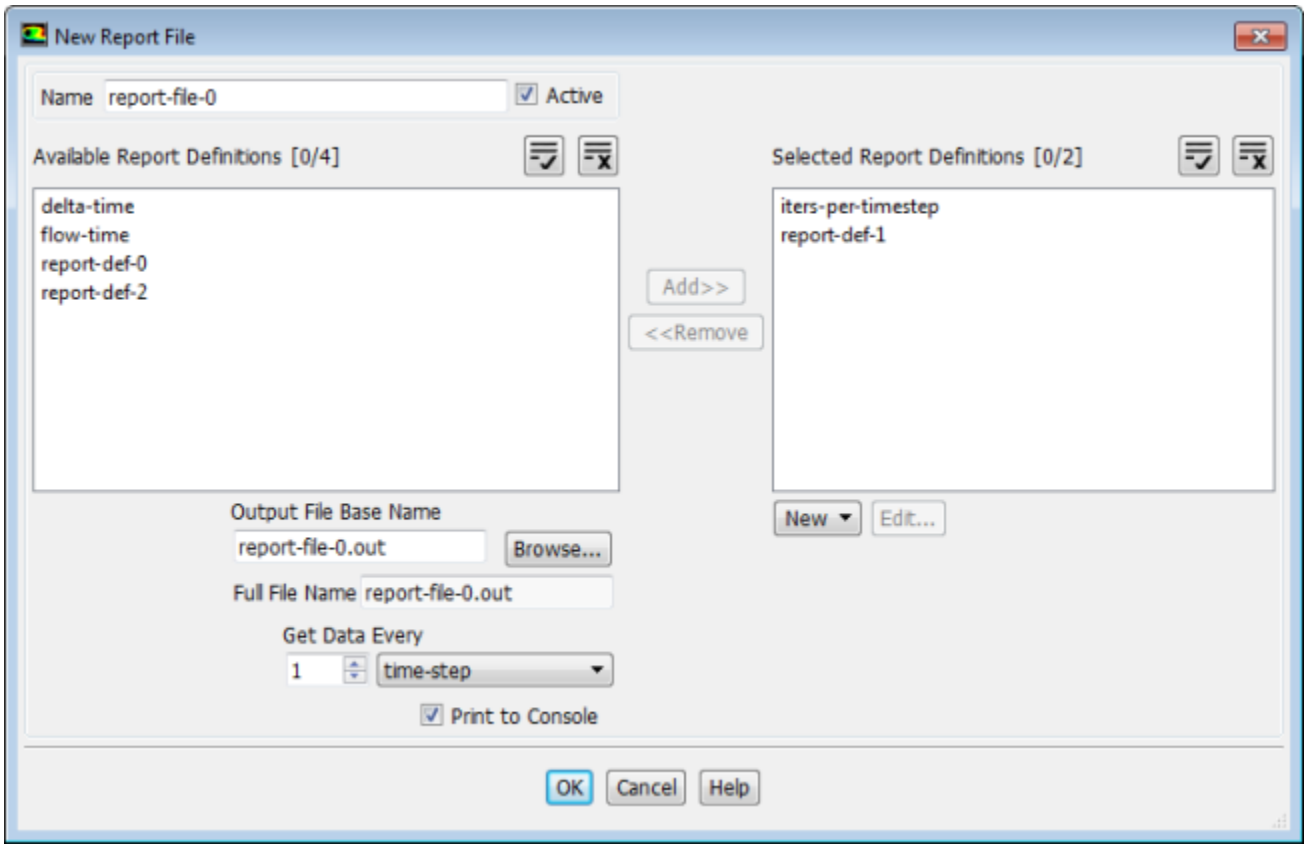
Summary
displays a summary of the saved motion definitions in the console.

Write...
allows you to write your saved motion definitions to an ASCII text file. See Mesh Morpher/Optimizer File Formats (p. 2867) for further details about the text file format.

38.2.57. New Report File Dialog Box

Solving → Reports → File...

Click **New...** in the **Report File Definitions** dialog box to open the **New Report File** dialog box.



Controls

Name
editable field displaying the name of the report file.

Active
check box (enabled by default) that determines if the file is written during the solution calculation.

Available Report Definitions
lists all of the existing report definitions not currently part of this report file.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Add>>
moves the selected report definition(s) to the **Selected Report Definitions** list.

<<Remove
moves the selected report definition(s) to the **Available Report Definitions** list.

Output File Base Name
allows you to specify the name used when the file is saved.

Browse...
opens the **Select File** dialog box, allowing you to specify the location where the report file will be saved.

Full File Name
displays the name and location of the report file once it has been saved.

Get Data Every
indicates the frequency that the report definition data is written to the file. The default value of 1 allows you to write to the file at every iteration or time-step. Time-step is only a valid choice you are calculating unsteady flow.

Print to Console
indicates whether or not the contents of the report file are printed to the console at the frequency specified in the **Get Data Every** field.

Selected Report Definitions
lists all of the existing report definitions currently included in this report file.

New
drop-down list allowing you to create new report definitions (surface, volume, force, flux, dpm, and user-defined).

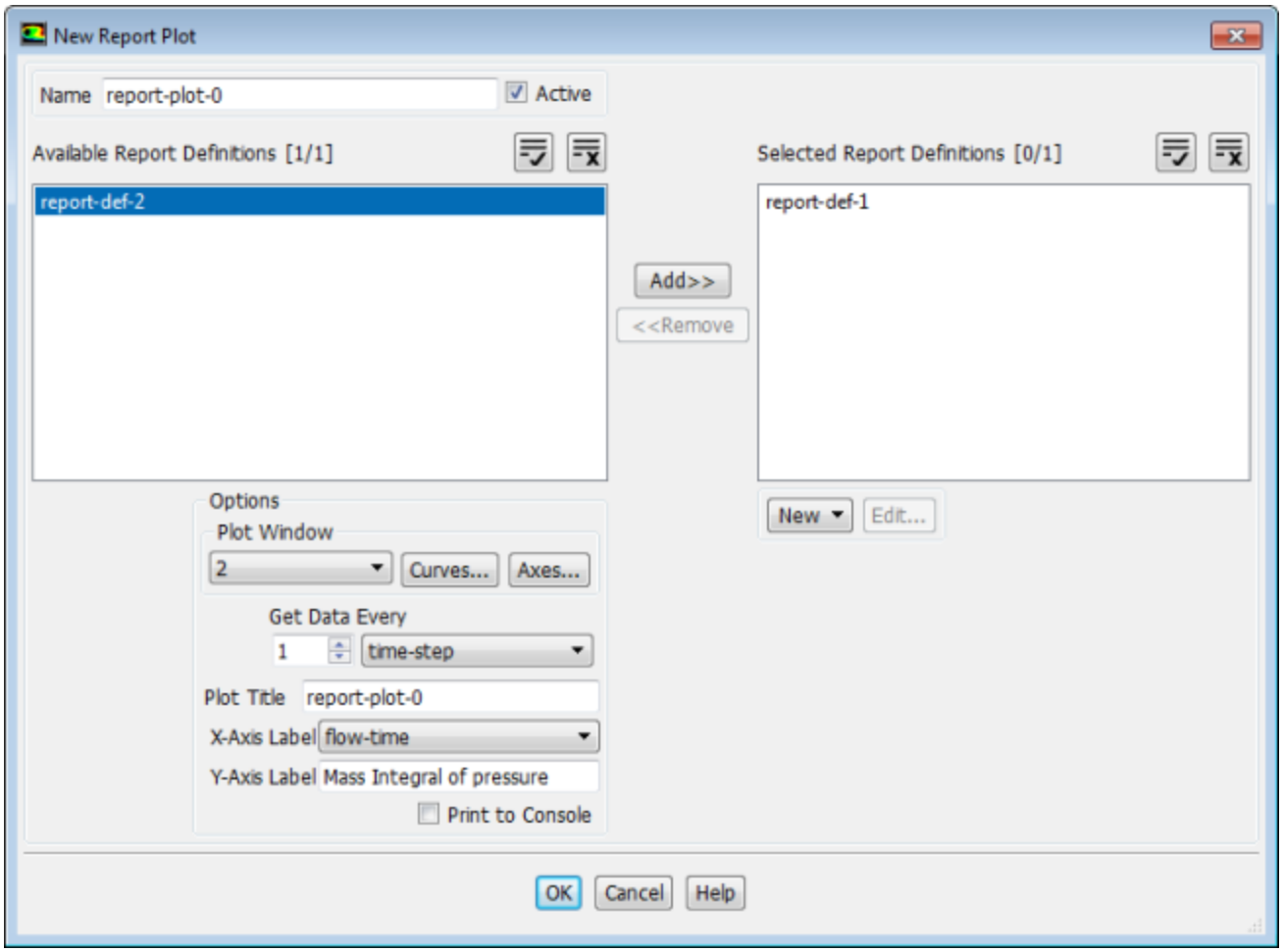
Edit...
allows you to modify the highlighted report definition in the **Selected Report Definition** list.

38.2.58. New Report Plot Dialog Box

Solving → Reports → Plot...

Click **New...** in the **Report Plot Definitions** dialog box to open the **New Report Plot** dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name
editable field displaying the name of the report plot.

Active
check box (enabled by default) that determines if the plot is plotted during the solution calculation.

Available Report Definitions
lists all of the existing report definitions not currently part of this report plot.

Add>>
moves the selected report definition(s) to the **Selected Report Definitions** list.

<<Remove
moves the selected report definition(s) to the **Available Report Definitions** list.

Options

Plot Window
allows you to specify the graphics window for this report plot.

Curves...
opens the Curves Dialog Box (p. 2610).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Axes...
opens the Axes Dialog Box (p. 2608).

Get Data Every
indicates the frequency that the report definition data is plotted in the graphics window. The default value of 1 allows you to plot the data at every iteration or time-step. Time-step is only a valid choice you are calculating unsteady flow.

Plot Title
provide a title for the plot.

X-Axis Label
choose whether the X-axis is flow-time or time-step.

Y-Axis Label
enter a label for the Y-axis.

Print to Console
indicates whether or not the contents of the report plot are printed to the console at the frequency specified in the **Get Data Every** field.

Selected Report Definitions
lists all of the existing report definitions currently included in this report plot.

New
drop-down list allowing you to create new report definitions (surface, volume, force, flux, dpm, and user-defined).

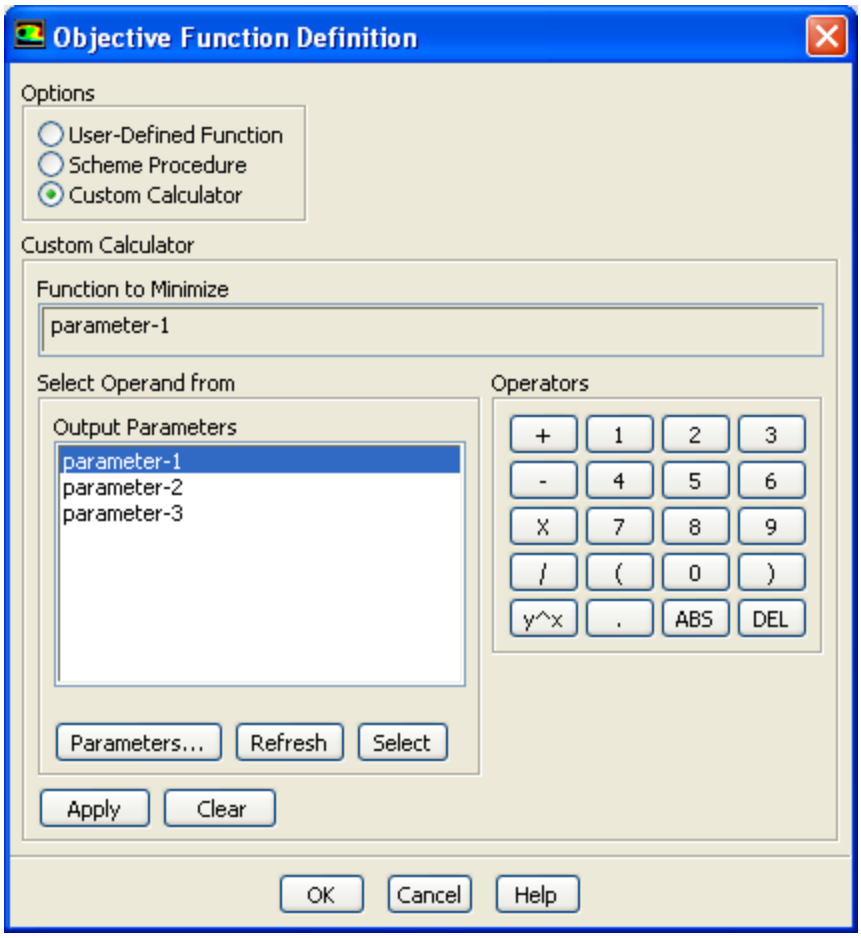
Edit...
allows you to modify the highlighted report definition in the **Selected Report Definition** list.

38.2.59. Objective Function Definition Dialog Box

The **Objective Function Definition** dialog box allows you to specify the format of the objective function that will be minimized by the mesh morpher/optimizer and (when it is a customized function of output parameters) to define the objective function. See Using the Mesh Morpher/Optimizer (p. 2125) for details about using this dialog box.

The **Objective Function Definition** dialog box is opened from the Mesh Morpher/Optimizer Dialog Box (p. 2734).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options
contains options for the format of the objective function.

User-Defined Function
specifies that the objective function is provided via a user-defined function.

Scheme Procedure
specifies that the objective function is provided via a Scheme source file. This option is not available if **newuoa** is selected from the **Optimizer** drop-down list in the **Optimizer** tab of the **Mesh Morpher/Optimizer** dialog box.

Custom Calculator
specifies that the objective function is based on output parameters, and is defined by the controls in the **Custom Calculator** group box. This option is not available if **newuoa** is selected from the **Optimizer** drop-down list in the **Optimizer** tab of the **Mesh Morpher/Optimizer** dialog box.

Custom Calculator
allows you to define the objective function. This group box is only available when **Custom Calculator** is selected from the **Options** list.

Function to Minimize
displays the objective function defined by the controls in the **Custom Calculator** group box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select Operand from
allows you to include output parameters in the definition of the objective function.

Output Parameters
provides a list of available output parameters, which can be included in the definition of the objective function.

Parameters...
opens the Parameters Dialog Box (p. 2627), which you can use to create additional output parameters.

Refresh
updates the **Output Parameters** list, so that all available output parameters are displayed.

Select
enters the output parameter that is currently selected in the **Output Parameters** list in the **Function to Minimize** field.

Operators
allows you to include calculator operators in the definition of the objective function. Clicking a button in this group box causes the appropriate symbol to appear in the **Function to Minimize** field.

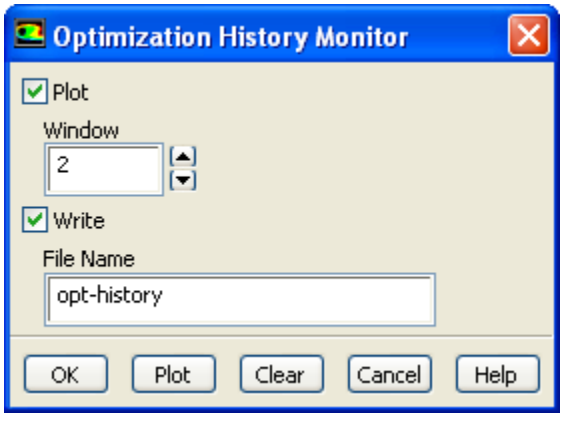
Apply
saves the custom objective function (displayed in the **Function to Minimize** field) as part of the case file.

Clear
deletes all of the text in the **Function to Minimize** field.

38.2.60. Optimization History Monitor Dialog Box

The **Optimization History Monitor** dialog box allows you to plot and/or record optimization history data, that is, how the value of the objective function (defined using the Mesh Morpher/Optimizer Dialog Box (p. 2734)) varies with each design stage produced by the mesh morpher/optimizer. See Using the Mesh Morpher/Optimizer (p. 2125) for details about using this dialog box.

The **Optimization History Monitor** dialog box is opened from the Mesh Morpher/Optimizer Dialog Box (p. 2734).



Controls

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plot
enables the plotting of the optimization history data in the graphics window (with the ID specified in **Window**).

Window
sets the ID of the graphics window in which the plot will be displayed. This number-entry box is only available when the **Plot** option is enabled. While ANSYS Fluent is iterating, the active graphics window is temporarily set to this ID to update the optimization history plot, and then it is returned to its previous value. Thus, the optimization history plot can be maintained in a separate window that does not interfere with other graphical postprocessing.

Write
enables the saving of the optimization history data to a file (with the name specified in **File Name**).

File Name
specifies the name of the file to which the optimization history data is written. This text-entry box is only available when the **Write** option is enabled.

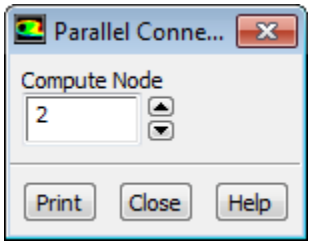
Plot
displays an XY plot of the optimization history data generated during the last calculation. Note that no plot will be displayed if the data was discarded using the **Clear** button.

Clear
discards the optimization history data, including the associated files. Each of these actions must be confirmed in a Question Dialog Box (p. 18).

38.2.61. Parallel Connectivity Dialog Box

The **Parallel Connectivity** dialog box prints the connectivity of the selected compute node. See Checking Network Connectivity (p. 2059) for details.

Parallel → Network → Connectivity...



Controls

Compute Node
indicates the compute node ID for which connectivity information is desired.

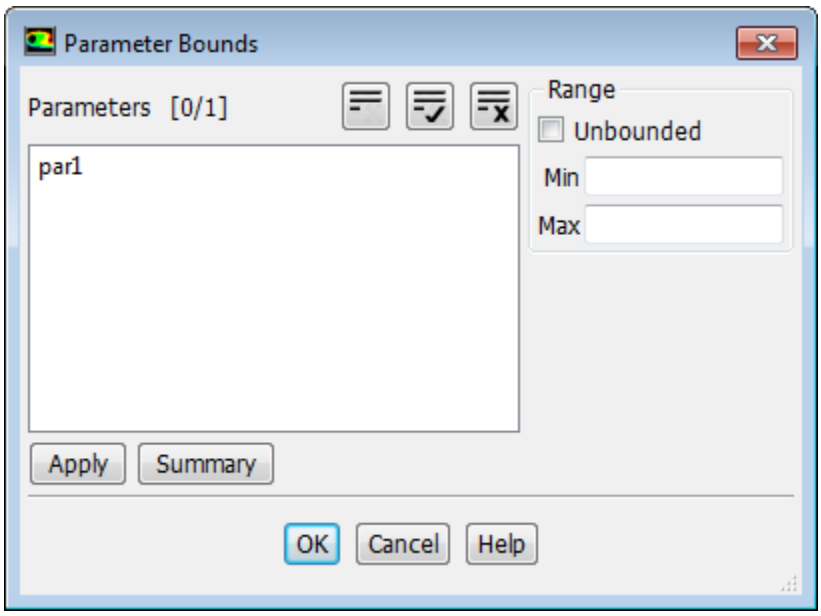
Print
prints information (in the console window) about the network connectivity for the selected compute node.

38.2.62. Parameter Bounds Dialog Box

The **Parameter Bounds** dialog box allows you to limit how much each parameter is allowed to deform when using the mesh morpher/optimizer. See Using the Mesh Morpher/Optimizer (p. 2125) for details about using this dialog box.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Parameter Bounds** dialog box is opened from the Mesh Morpher/Optimizer Dialog Box (p. 2734).



Controls

Parameters
selects the parameters (created using the **Deformation** tab of the Mesh Morpher/Optimizer Dialog Box (p. 2734)) for which you are defining bounds.

Range
allows you to specify the bounds.

Unbounded
allows you to specify whether the selected **Parameters** are unbounded.

Min, Max
allows you to set the minimum and maximum values allowed for the selected **Parameters** when the **Unbounded** option is disabled.

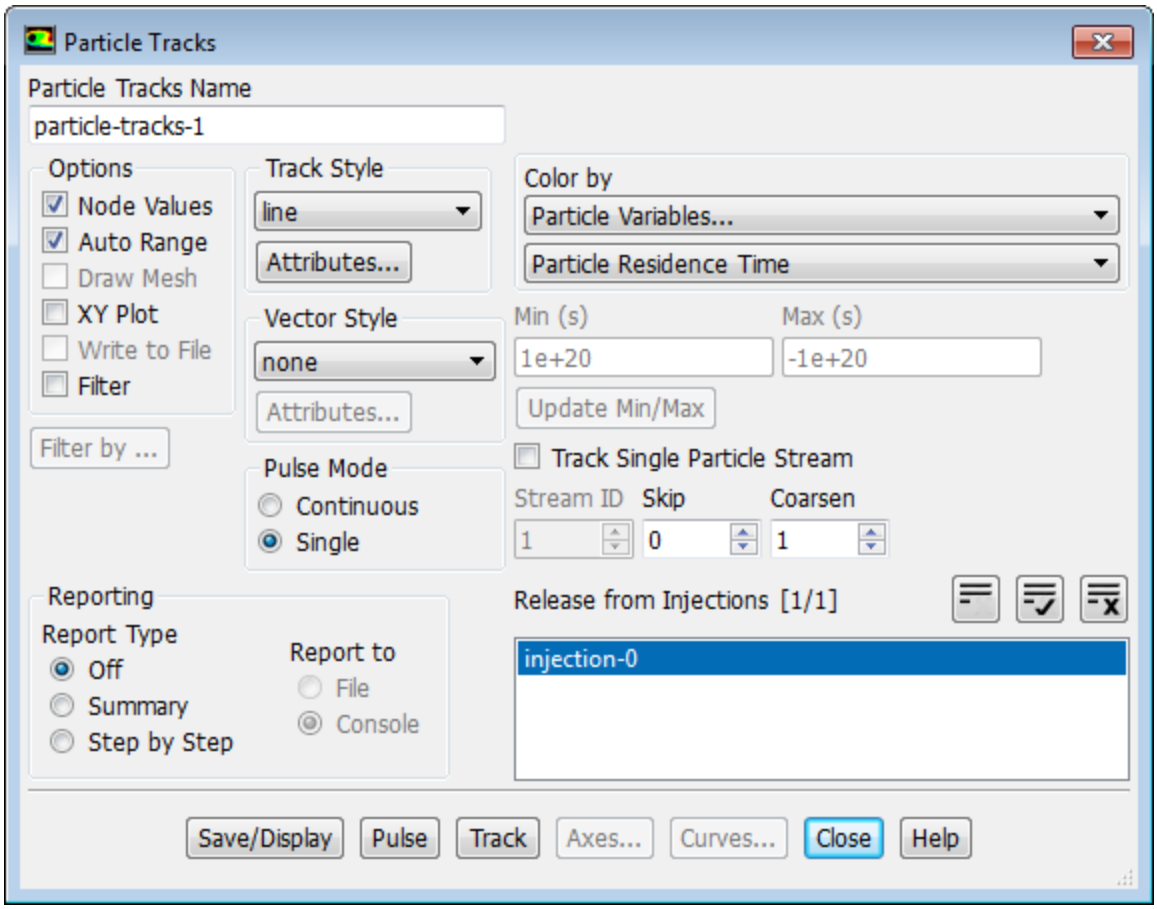
Apply
saves the bounds for the currently selected **Parameters**.

Summary
displays a summary of all of the saved parameter bounds in the console.

38.2.63. Particle Tracks Dialog Box

The **Particle Tracks** dialog box controls the generation and display of discrete phase particles. See Displaying of Trajectories (p. 1330) for details about the items below.

Postprocessing → Graphics → Particle Tracks → New...



Controls

Particle Tracks Name

is the name for a particle tracks definition. You can specify a name or use the default name **`particle-tracks-`*`id`***. This control appears only for particle tracks definitions.

Options

contains the options described below:

Node Values

specifies that node values should be interpolated to compute the scalar field at a particle location. This option is turned on by default. See Choosing Node or Cell Values (p. 1780) for details.

Auto Range

specifies that the minimum and maximum values of the scalar field will be the limits of that field. If this option is not selected, you can enter **Min** and **Max** values manually. These values determine the range of the color scale.

Draw Mesh

toggles between displaying and not displaying the mesh. The Mesh Display Dialog Box (p. 2163) is opened when **Draw Mesh** is selected.

XY Plot

enables the display of an XY plot along the particle trajectories. When this option is selected, the **Display** push button will change to **Plot**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Write to File
(available only when **XY Plot** is on) activates the file-writing option. When this option is selected, the **Plot** push button will change to **Write...**. Clicking on the **Write...** button will open The Select File Dialog Box (p. 19), in which you can specify a name and save a file containing the XY plot data. The format of this file is described in XY Plot File Format (p. 1850).

Enable Filter
when activated allows you to click the **Filter by...**.

Filter by...
opens the Particle Filter Attributes (p. 2561) where you can specify how you would like to filter the particles being displayed.

Reporting
allows you to specify how you would like to report particle tracks.

Report Type
controls the type of trajectory-fate reports to be displayed.

Off
disables reporting of trajectory fates.

Summary
enables summary reports of trajectory fates. See Reporting of Trajectory Fates (p. 1340) for details.

Step by Step
enables the step-by-step reporting of trajectories. This item will not appear if you have requested unsteady tracking in the Discrete Phase Model Dialog Box (p. 2278). See Step-by-Step Reporting of Trajectories (p. 1347) for details.

Current Positions
enables the reporting of the positions and velocities of all particles that are in the domain at the current time. This item appears only when unsteady tracking has been requested in the Discrete Phase Model Dialog Box (p. 2278). See Reporting of Current Positions for Unsteady Tracking (p. 1349) for details.

Report to
indicates the destination of the trajectory report.

File
enables the writing of the trajectory report to a file. When this option is enabled, the **Track** button will become the **Write...** button.

Console
enables the display of the trajectory report information in the console window.

Significant Figures
controls the number of significant figures used when a **Step by Step** or **Current Positions** report is selected.

Reporting Variables...
opens the Reporting Variables Dialog Box (p. 2562) that contains a list from which you can select the scalar field to be used to color the particle tracks.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Track Style
sets the trajectory style. Trajectories can be displayed as lines, ribbons, spheres, cylinders, or a set of points. Pulsing can be done only on **point** or **line** styles.

Attributes...
opens the Track Style Attributes Dialog Box (p. 2562), Particle Sphere Style Attributes Dialog Box (p. 2563), or the Ribbon Attributes Dialog Box (p. 2560), depending on the selected **Track Style**. These dialog boxes allow you to set the attributes for the chosen trajectory style.

Vector Style
sets the trajectory vector style. You can display the trajectories vectors in three different ways: **vector**, **centered-vector**, and **centered-cylinder**. All three styles are demonstrated in Controlling the Vector Style of Particle Tracks (p. 1334).

Attributes...
opens the Particle Vector Style Attributes Dialog Box (p. 2564).

Pulse Mode
specifies either a single or continuous release of particles that follow the trajectories, starting at the release surface, when you use the **Pulse** button to animate the trajectories.

Continuous
sets the pulse mode to continuously release particles from the initial positions.

Single
sets the pulse mode to release a single wave of particles.

The **Pulse Mode** is used only when particles are being **Pulsed**.

Y Axis Function
contains a list of solution variables that can be used for the y axis of the plot.

This item appears when the **XY Plot** option is turned on.

X Axis Function
contains a list of functions for the X-axis of the plot. You can plot the quantity selected in the **Y Axis Function** drop-down list as a function of the **Time** elapsed along the trajectory, or the **Path Length** along the trajectory.

This item appears when the **XY Plot** option is turned on.

Min/Max
display the values to which the lower and upper ends of the color scale map. If you are using the **Auto Range** option, these values will be automatically computed when you click **Compute**. If you are not using **Auto Range**, you can enter values manually.

Update Min/Max
computes the **Min** and **Max** for the field chosen in **Color by**.

Track PDF Transport Particles
enables the tracking of PDF transport particles. This item is available only when the composition PDF transport model is enabled.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Free Stream Particles
enables the tracking of free stream particles. Note that this option is available only when the Lagrangian wallfilm model is enabled in the wall boundary conditions dialog box, under the **DPM** tab.

Wall Film Particles
enables the tracking of wall film particles. Note that this option is available only when the Lagrangian wallfilm model is enabled in the wall boundary conditions dialog box, under the **DPM** tab.

Track Single Particle Stream
enables the tracking of a single particle stream in the selected injection, instead of all the streams in that injection. See Specifying Particles for Display (p. 1332) for details.

Stream ID
specifies the particle stream to be tracked when the **Track Single Particle Stream** option is enabled.

Skip
allows you to "thin" or "sample" the particles that are displayed.

Coarsen
reduces the plotting time by reducing the number of points that are plotted for a given trajectory in any cell. This is valid only for steady-state cases.

Release from Injections
selects the injections from which to release particles. You must select at least one injection before any particles can be released.

Display
displays the trajectories and generates a trajectory report in the console window (if requested).

Save/Display
displays the particle tracks in the active graphics window and saves the particle track definition. This button appears only for particle track definitions and replaces the **Display** button.

Plot
displays an XY plot along the particle trajectories. This button replaces the **Display** button when the **XY Plot** option is turned on.

Write...
opens The Select File Dialog Box (p. 19), in which you can save the XY plot data or trajectory report to a file. This button replaces the **Plot** button when the **Write to File** option is turned on, or the **Track** button when the **File** option is selected under **Report to**.

Pulse
animates the position of particles. If **Pulse Mode** is set to **Continuous**, particles are reintroduced at the seed positions repeatedly. If the mode is **Single**, a single wave of particles moves through the domain.

The label of this button changes to **Stop !** when the animation is in progress. Click **Stop !** to stop pulsing (and the label goes back to **Pulse**).

Track
computes the particle trajectories and generates any reports without displaying the trajectories in the graphics window.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

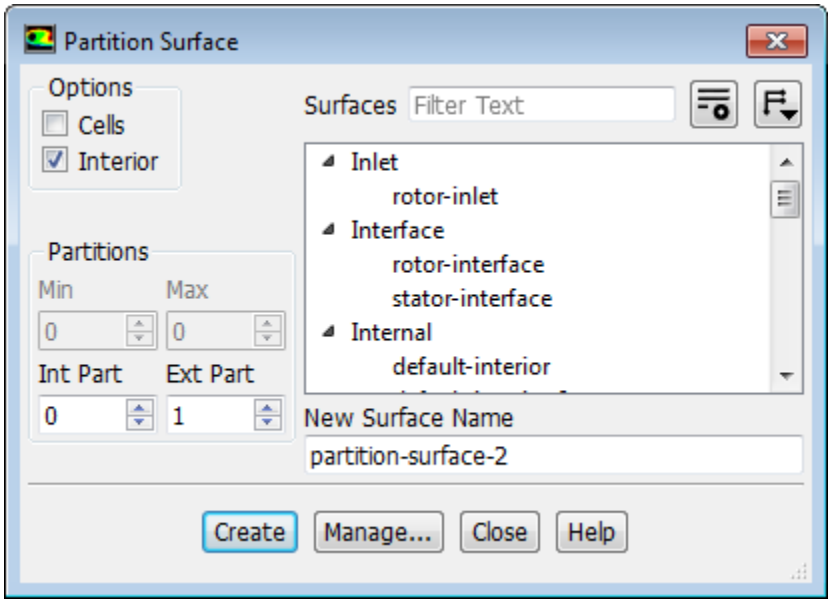
Axes...
opens the Axes Dialog Box (p. 2608), which allows you to customize the XY plot axes. This item appears when the **XY Plot** option is turned on.

Curves...
opens the Curves Dialog Box (p. 2610), which allows you to customize the curves used in the XY plot. This item appears when the **XY Plot** option is turned on.

38.2.64. Partition Surface Dialog Box

The **Partition Surface** dialog box allows you to create a surface defined by the boundary of two adjacent mesh partitions. See Partition Surfaces (p. 1712) for details about the items below.

Setting Up Domain → Surface → Create → Partition...



Controls

Options
contains check buttons that allow you to choose interior or exterior faces or cells to be contained in the partition surface.

Cells
specifies, when enabled, that the partition surface will consist of cells (either interior or exterior, depending on the status of the **Interior** check button) that lie on the partition boundary. When this option is disabled, the faces on the boundary between partitions will make up the partition surface. Depending on the status of the **Interior** check button, the faces will reflect data values for the interior or exterior cells.

Interior
toggles between interior and exterior cells. If the partition surface consists of faces instead of cells, **Interior** specifies for which cells data will be displayed on the faces. When this option is enabled, interior cells (cells that are on the **Int Part** side of the partition boundary) will make up the partition surface. When it is disabled, the partition surface will consist of exterior cells (cells that are on the **Ext Part** side of the partition boundary).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Partitions
contains information about which partitions are under consideration. The boundary that defines the partition surface is the boundary between the **Int Part** and the **Ext Part**. If there are more than two mesh partitions, each interior partition will share boundaries with several exterior partitions. By setting the appropriate values for **Int Part** and **Ext Part**, you can create surfaces for any of these boundaries.

Min/Max
indicates the minimum and maximum ID numbers of the mesh partitions. The minimum is always zero, and the maximum is one less than the number of processors.

Int Part
indicates the ID number of the interior partition (that is, the partition under consideration).

Ext Part
indicates the ID number of the bordering partition.

Surfaces
displays an informational list of existing surfaces.

New Surface Name
designates the name of the new surface. The default is the concatenation of the surface type and an integer which is the new surface ID.

Create
creates the surface.

Manage...
opens the Surfaces Dialog Box (p. 2810) in which you can rename and delete surfaces and determine their sizes.

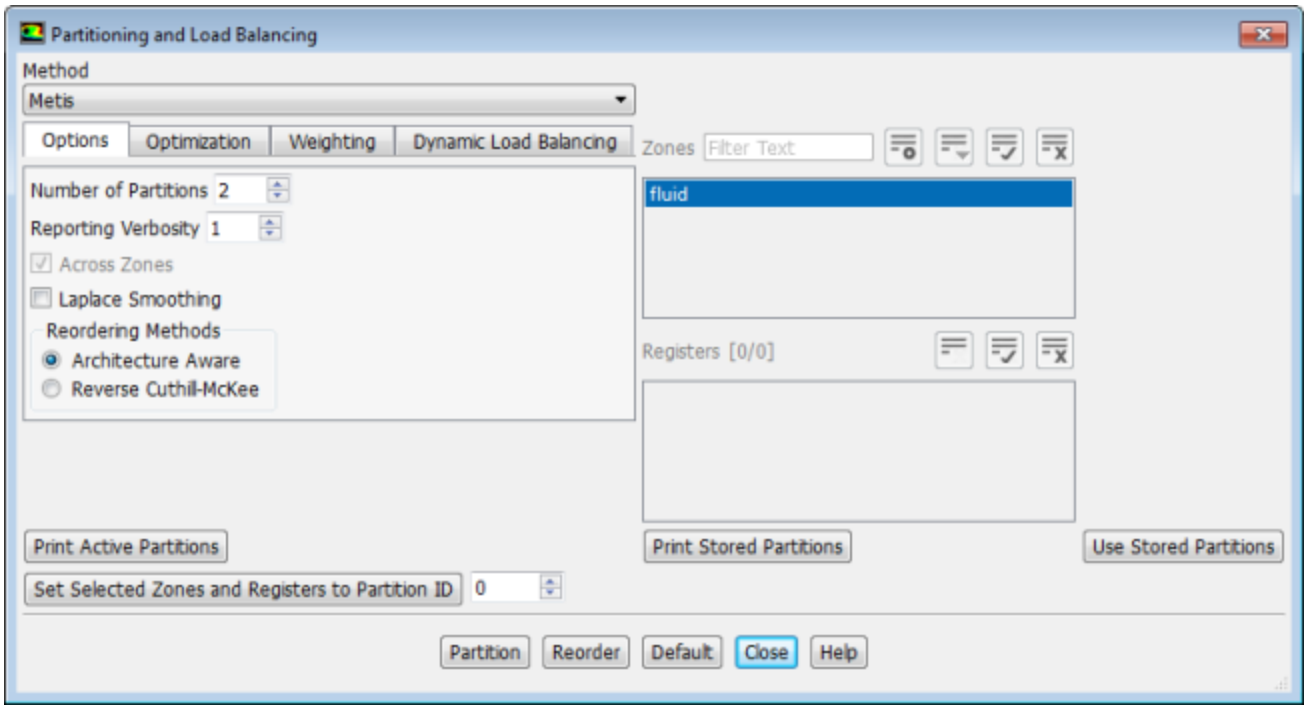
38.2.65. Partitioning and Load Balancing Dialog Box

The **Partitioning and Load Balancing** dialog box allows you to partition the mesh into separate clusters of cells for separate processors on a parallel computer. Note that the **Partitioning and Load Balancing** dialog box is slightly different in the serial and parallel solvers. See Partitioning the Mesh Manually and Balancing the Load (p. 2033) for details.

The **Partitioning and Load Balancing** dialog box also allows you to enable and control ANSYS Fluent's load balancing feature. Load balancing will automatically detect and analyze parallel performance, and redistribute cells between the existing compute nodes to optimize it. See Load Balancing (p. 2044) for more information about load balancing.

Parallel → General → Partitioning/Load Balancing...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Method

contains a drop-down list of the recursive partition methods that can be used to create the mesh partitions. The choices include the **Cartesian Axes**, **Cartesian Strip**, **Cartesian X-Coordinate**, **Cartesian Y-Coordinate**, **Cartesian Z-Coordinate**, **Cartesian R Axes**, **Cartesian RX-Coordinate**, **Cartesian RY-Coordinate**, **Cartesian RZ-Coordinate**, **Cylindrical Axes**, **Cylindrical R-Coordinate**, **Cylindrical Theta-Coordinate**, **Cylindrical Z-Coordinate**, **Metis**, **Polar Axes**, **Polar R-Coordinate**, **Polar Theta-Coordinate**, **Principal Axes**, **Principal Strip**, **Principal X-Coordinate**, **Principal Y-Coordinate**, **Principal Z-Coordinate**, **Spherical Axes**, **Spherical Rho-Coordinate**, **Spherical Theta-Coordinate**, and **Spherical Phi-Coordinate** techniques, which are described in Mesh Partitioning Methods (p. 2046).

Options

tab

Number of Partitions

defines the desired number of mesh partitions. This usually matches the number of processors available for parallel computing.

Reporting Verbosity

specifies the amount of information to be reported in the text (console) window during the partitioning. With the default value of 1, Fluent will print the number of partitions created, the number of bisections performed, the time required for the partitioning, and the minimum and maximum cell, face, interface, and face-ratio variations. (See Interpreting Partition Statistics (p. 2052) for details.) If you increase the **Reporting Verbosity** to 2, the partition method used, the partition ID, number of cells, faces, and interfaces, and the ratio of interfaces to faces for each partition will also be printed in the console window. If you decrease the **Reporting Verbosity** to 0, only the number of partitions created and the time required for the partitioning will be reported.

Across Zones

allows partitions to cross zone boundaries (the default). If turned off, it will restrict partitioning to within each cell zone. This is recommended *only* when cells in different zones require significantly

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

different amounts of computation during the solution phase, for example if the domain contains both solid and fluid zones.

Laplace Smoothing
enables the Laplace smoothing method, which can be used to prevent partition boundaries from lying along regions of high-aspect ratio cells. This can improve convergence behavior in cases with highly stretched cells.

Cutoff Aspect Ratio
controls the threshold for the Laplace smoothing method.

Reordering Method
is used to optimize parallel performance

Architecture Aware
is the default option and it accounts for the system architecture and network topology in remapping the partitions to the processors.

Reverse Cuthill-McKee
minimizes the bandwidth of the compute-node connectivity matrix (the maximum distance between two connected processes) without incorporating the system architecture.

Optimization
tab contains options for enabling schemes to optimize the partitions created by the selected partition method. In addition, the optimization scheme will be applied until appropriate criteria are met, or the maximum number of iterations have been run. If the **Iterations** counter is set to 0, the optimization scheme will be applied until completion, without limit on the maximum number of iterations.

Merge
attempts to decrease the number of interfaces by eliminating orphan cell clusters (an orphan cluster is a group of connected cells whose members each have at least one face coincident with an interface boundary).

Smooth
attempts to minimize the number of interfaces by sacrificing cells on the partition boundary to the neighboring partition to reduce the partition boundary surface area.

Pre-Test
instructs Fluent to test all coordinate directions and choose the one which yields the fewest partition interfaces for the final bisection. Note that this option is available only when you choose **Principal Axes** or **Cartesian Axes** as the partitioning method.

Weighting
tab allows you to set the appropriate weights, prior to partitioning the mesh.

Weight Types
allows you to turn on/off a specific weight. Weight types include: **Faces per Cell** (which is enabled by default, with additional weighting of **2**), **Solid Zones** (available only if solid cell zones are defined); **VOF** (available only if the volume of fluid multiphase model is turned on); **DPM** (available only for discrete phase simulations with injections defined), where you can specify **Hybrid Optimization**; and **ISAT**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

User-Specified
allows you to use the default value (by not selecting the check box), or to specify a value yourself (by selecting the check box and entering a numerical value in the **Value** field.

Value
allows you to enter a user-specified numerical value that either defines the ratio used in the weighting, or (for the **Faces per Cell** weighting) is added to the calculated weight of each cell.

Dynamic Load Balancing
tab allows you to set load balancing thresholds and intervals.

Physical Models
allows you to set thresholds and intervals for the simulation's physical models.

Threshold
is a percentage and depending on the value, load balancing will occur if the load imbalance is greater than the value entered.

Interval
allows you to specify the desired interval for load balancing cycles, in terms of number of iterations. When a value of 0 is specified, ANSYS Fluent will internally determine the best value to use, initially using an interval of 25 iterations. You can override this behavior by specifying a nonzero value. ANSYS Fluent will then attempt to perform load balancing after every N iterations, where N is the specified **Balance Interval**.

Dynamic Mesh
allows you to set load balancing threshold and interval values for your dynamic mesh. This option is only available when the dynamic mesh model is enabled.

Auto
allows a percentage of interface faces and loads to be automatically traced.

Threshold
is a percentage and depending on the value, load balancing will occur if the load imbalance is greater than the value entered.

Interval
allows you to specify the desired interval for load balancing cycles, in terms of number of iterations. When a value of 0 is specified, ANSYS Fluent will internally determine the best value to use, initially using an interval of 25 iterations. You can override this behavior by specifying a nonzero value. ANSYS Fluent will then attempt to perform load balancing after every N iterations, where N is the specified **Balance Interval**.

Mesh Adaption
allows you to set threshold values for mesh adaption.

Threshold
is a percentage and depending on the value, load balancing will occur if the load imbalance is greater than the value entered.

Zones
contains a list of cell zones. Partitioning will be applied to cells in zones selected from this list.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Registers
contains a list of cell registers that have been created using the adaption tools. You can restrict partitioning to a group of cells by selecting a register containing the cells. See Partitioning Within Zones or Registers (p. 2043) for details.

Print Active Partitions
(parallel solver only) prints the partition ID, number of cells, faces, and interfaces, and the ratio of interfaces to faces for each active partition (see Checking the Partitions (p. 2052) for information about active partitions) in the console window. In addition, it prints the minimum and maximum cell, face, interface, and face-ratio variations.

Print Stored Partitions
(parallel solver only) prints the partition ID, number of cells, faces, and interfaces, and the ratio of interfaces to faces for each stored partition (see Checking the Partitions (p. 2052) for information about stored partitions) in the console window. In addition, it prints the minimum and maximum cell, face, interface, and face-ratio variations.

Print Partitions
(serial solver only) prints the partition ID, number of cells, faces, and interfaces, and the ratio of interfaces to faces for each stored partition in the console window. In addition, it prints the minimum and maximum cell, face, interface, and face-ratio variations.

Set Selected Zones and Registers to Partition ID
allows you to set a value that assigns selected **Zones** and/or **Registers** to a specific partition ID. A region or zone is marked before setting the marked cells to one of the partition IDs.

Use Stored Partitions
(parallel solver only) allows you to make the stored cell partitions the active cell partitions. The active cell partition is used for the current calculation, while the stored cell partition (the last partition performed) is used when you save a case file.

Partition
subdivides the mesh into the selected number of partitions using the prescribed method and optimization(s).

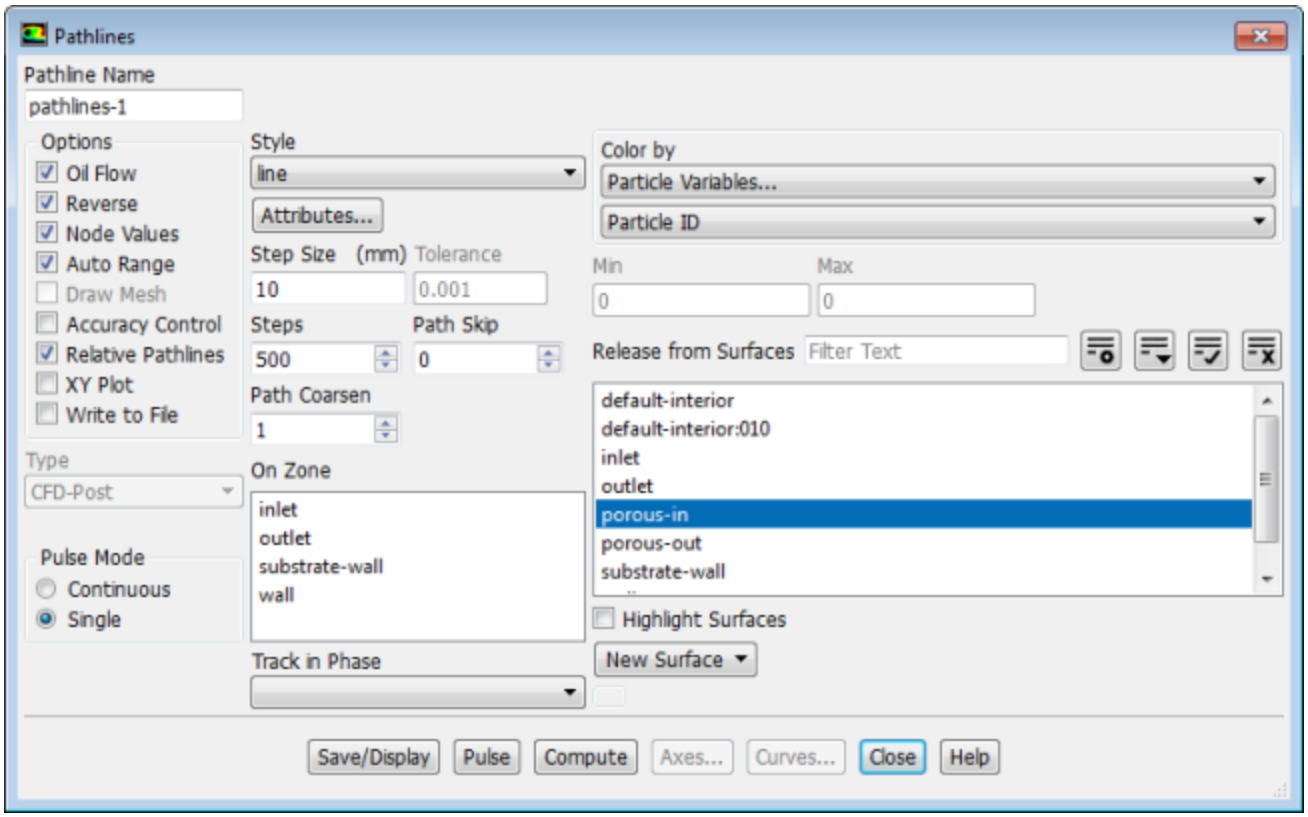
Default
sets all controls to their default values, as assigned by ANSYS Fluent. After execution, the **Default** button becomes the **Reset** button.

Reset
resets the fields to their most recently saved values (that is, the values before **Default** was selected). After execution, the **Reset** button becomes the **Default** button.

38.2.66. Pathlines Dialog Box

The **Pathlines** dialog box controls the generation and display of pathlines. See Displaying Pathlines (p. 1772) for details about the items below.

Postprocessing → **Graphics** → **Pathlines** → **New...**



Controls

Pathline Name
is the name for a pathline definition. You can specify a name or use the default name **`pathline-`*`id`***. This control appears only for pathline definitions.

Options
contains the options described below:

Oil Flow
toggles between regular pathlines and oil-flow pathlines. When this option is selected, pathlines are constrained to lie on the zone selected in the **On Zone** list.

Reverse
specifies that each particle's path is traced back in time. This option is turned off by default.

Node Values
specifies that node values should be interpolated to compute the scalar field at a particle location. This option is turned on by default. See Choosing Node or Cell Values (p. 1780) for details.

Auto Range
specifies that the minimum and maximum values of the scalar field will be the limits of that field. If this option is not selected, you can enter **Min** and **Max** values manually. These values determine the range of the color scale.

Draw Mesh
when enabled opens the Mesh Display Dialog Box (p. 2163) where you can specify how to display your mesh.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Accuracy Control
allows you to specify tolerance to control the pathlines accuracy.

Relative Pathlines
allows you to display the pathlines relative to the rotating reference frame.

XY Plot
enables the display of an XY plot along the pathline trajectories. When this option is selected, the **Display** push button will change to **Plot**.

Write to File
activates the file-writing option. When this option is selected, the **Type** option becomes active and the **Plot** push button will change to **Write...**. Clicking on the **Write...** button will open The Select File Dialog Box (p. 19), in which you can specify a name and save a file containing the XY plot data. The format of this file is described in XY Plot File Format (p. 1850).

Type
specifies the type of the file you want to write.

CFD-Post
allows you to write the file in CFD-Post compatible format, which can be read in **CFD-Post**.

Fieldview
allows you to write the file in **FIELDVIEW** format, which can be read in **FIELDVIEW**.

Geometry
allows you to write the file in `.ibl` format, which can be read in GAMBIT.

EnSight
allows you to write the file in `.encas` format, which can be read in EnSight.

Pulse Mode
specifies either a single or continuous release of particles that follow the pathlines, starting at the release surface, when you use the **Pulse** button to animate the pathlines.

Continuous
sets the pulse mode to continuously release particles from the initial positions.

Single
sets the pulse mode to release a single wave of particles.

The **Pulse Mode** is used only when particles are being pulsed.

Style
sets the pathline style. Pathlines can be displayed as lines, ribbons, spheres, cylinders, or a set of points. The **ribbon** style also uses the **Twist By** field and the **TwistFactor** value (specified in the Ribbon Attributes Dialog Box (p. 2560)). Pulsing can be done only on **point** or **line** styles.

Attributes...
opens the Path Style Attributes Dialog Box (p. 2559) or the Ribbon Attributes Dialog Box (p. 2560), depending on the selected **Style**. These dialog boxes allow you to set the attributes (ribbon width, marker size, cylinder radius, twist-by field, and so on) for the chosen pathline style.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Step Size
sets the interval used for computing the next position of a particle. This is in units of length. Particle positions are always computed when they enter/leave a cell. If you specify a very large size, the particle positions at entry/exit of each cell will still be computed (and displayed).

Tolerance
allows you to control the error when using large time step sizes for the calculation. This is available when **Accuracy Control** is selected under **Options**.

Steps
sets the maximum number of steps a particle can advance. A particle's path may also end if it leaves the domain.

Path Skip
allows you to "thin" or "sample" the pathlines that are displayed. See "Thinning" Pathlines (p. 1775) for details.

Path Coarsen
reduces the time and the number of points plotted in a pathline. Coarsening factor of 'n' will result in plotting of each 'n'th point for a given pathline in each cell.

On Zone
contains a list from which you can select the zone on which the particles are constrained to lie. This list is activated only when the **Oil Flow** option is selected.

Track in Phase
(for the Eulerian Multiphase model only) allows you to select the phase or the mixture of all phases for tracking particle pathlines.

Color by
contains a list from which you can select the scalar field to be used to color the pathlines. The default is **Particle ID**, a unique ID for each particle.

Y Axis Function
contains a list of solution variables that can be used for the y axis of the plot.

This item appears when the **XY Plot** option is turned on.

X Axis Function
contains a list of functions for the x axis of the plot. You can plot the quantity selected in the **Y Axis Function** drop-down list as a function of the **Time** elapsed along the trajectory, or the **Path Length** along the trajectory.

This item appears when the **XY Plot** option is turned on.

Min/Max
display the values to which the lower and upper ends of the color scale map. If you are using the **Auto Range** option, these values will be automatically computed when you click **Compute**. If you are not using **Auto Range**, you can enter values manually.

Release from Surfaces
selects the surfaces from which to release particles. You must select at least one surface before any particles can be released.

Highlight Surfaces
when enabled highlights the surfaces (selected in the **Pathlines** dialog box) in the graphics window.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

New Surface
is a drop-down list button that contains a list of surface options:

Point
opens the Point Surface Dialog Box (p. 2780).

Line/Rake
opens the Line/Rake Surface Dialog Box (p. 2727).

Plane
opens the Plane Surface Dialog Box (p. 2777).

Quadric
opens the Quadric Surface Dialog Box (p. 2781).

Iso-Surface
opens the Iso-Surface Dialog Box (p. 2721).

Iso-Clip
opens the Iso-Clip Dialog Box (p. 2720).

EnSight Encas File Name
allows you to specify a.encas file name if you selected **EnSight** under **Type**.

Browse...
opens The Select File Dialog Box (p. 19) where you can select the `.encas` file.

Time Steps For EnSight Export
allows you to set the number of time levels that are available for animation in EnSight.

Display
displays the pathlines in the active graphics window.

Save/Display
plots the pathlines in the active graphics window and saves the pathline definition. This button appears only for pathline definitions and replaces the **Display** button.

Plot
displays an XY plot along the pathline trajectories. This button replaces the **Display** button when the **XY Plot** option is turned on.

Write...
opens The Select File Dialog Box (p. 19), in which you can save the XY plot data to a file. This button replaces the **Plot** button when the **Write to File** option is turned on.

Pulse
animates the position of particles. If **Pulse Mode** is set to **Continuous**, particles are reintroduced at the seed positions repeatedly. If the mode is **Single**, a single wave of particles moves through the domain.

The label of this button changes to **Stop !** when the animation is in progress. Click **Stop !** to stop pulsing (and the label goes back to **Pulse**).

Compute
computes the **Min** and **Max** for the field chosen in **Color by**.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

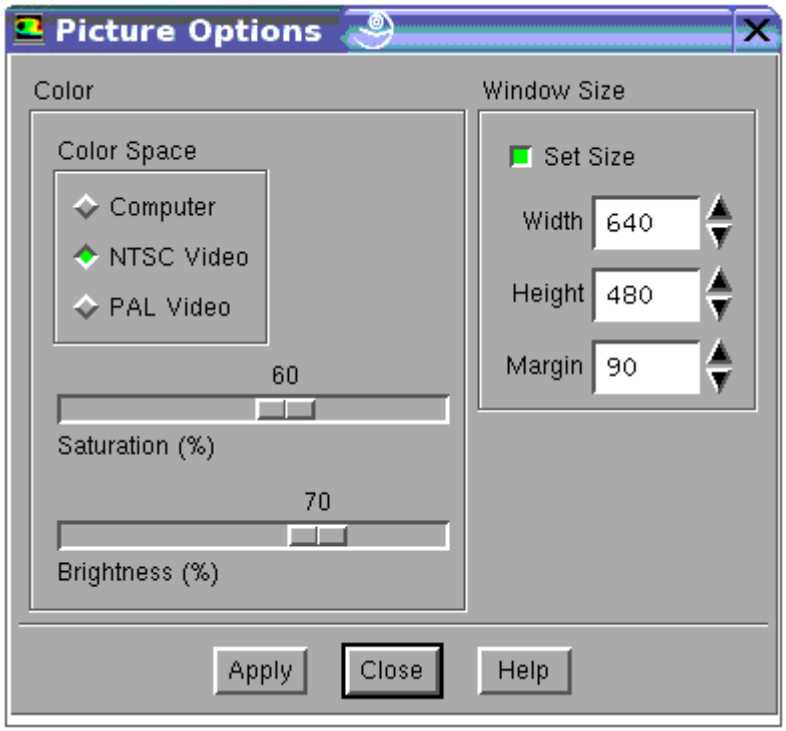
Axes...
opens the Axes Dialog Box (p. 2608), which allows you to customize the XY plot axes. This item appears when the **XY Plot** option is turned on.

Curves...
opens the Curves Dialog Box (p. 2610), which allows you to customize the curves used in the XY plot. This item appears when the **XY Plot** option is turned on.

38.2.67. Picture Options Dialog Box

The **Picture Options** dialog box allows you to adjust the color levels and size of the picture in the graphics window. This dialog box has an **Apply** button, so as you adjust levels, you can click **Apply** to see the effect on the picture in the current graphics window. Each option has a presetting, so you can click **Apply** immediately to see the effect of the presettings.

The **Picture Options** dialog box is opened from the Video Control Dialog Box (p. 2833).



Controls

Color
contains options for adjusting the color levels of the picture.

Color Space
lets you choose a filter that will restrict all colors to within a particular color range by modifying individual color values, if needed.

Computer
when selected, effectively turns off color filtering.

NTSC Video
when selected, restricts colors to the NTSC color space. NTSC (National Television Standards Committee) is the main video format used in North America and Japan.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PAL Video
when selected, restricts colors to the PAL color space. PAL (Phase Alternate Line) is the video format used by some countries in Western Europe, Asia, and Australia.

Saturation (%)
controls the percentage of color saturation (or purity) of the colors in the picture. A saturation in the range of 0-20% produces grayed-out colors. A saturation in the range of 40-60% produces pastel colors. A saturation in the range of 80-100% produces vivid colors. For video, this value should be no more than 80%.

Brightness (%)
controls the color brightness of the picture. When lowering the saturation, it is often desirable to lower the brightness a little bit as well to avoid pastel colors.

Window Size
allows you to resize the graphics window so that you can match up the size to your VTR device.

Set Size
lets you decide whether or not you want the **Width**, **Height**, and **Margin** values to be used to resize the graphics window.

Width
specifies the width in pixels of the new window size.

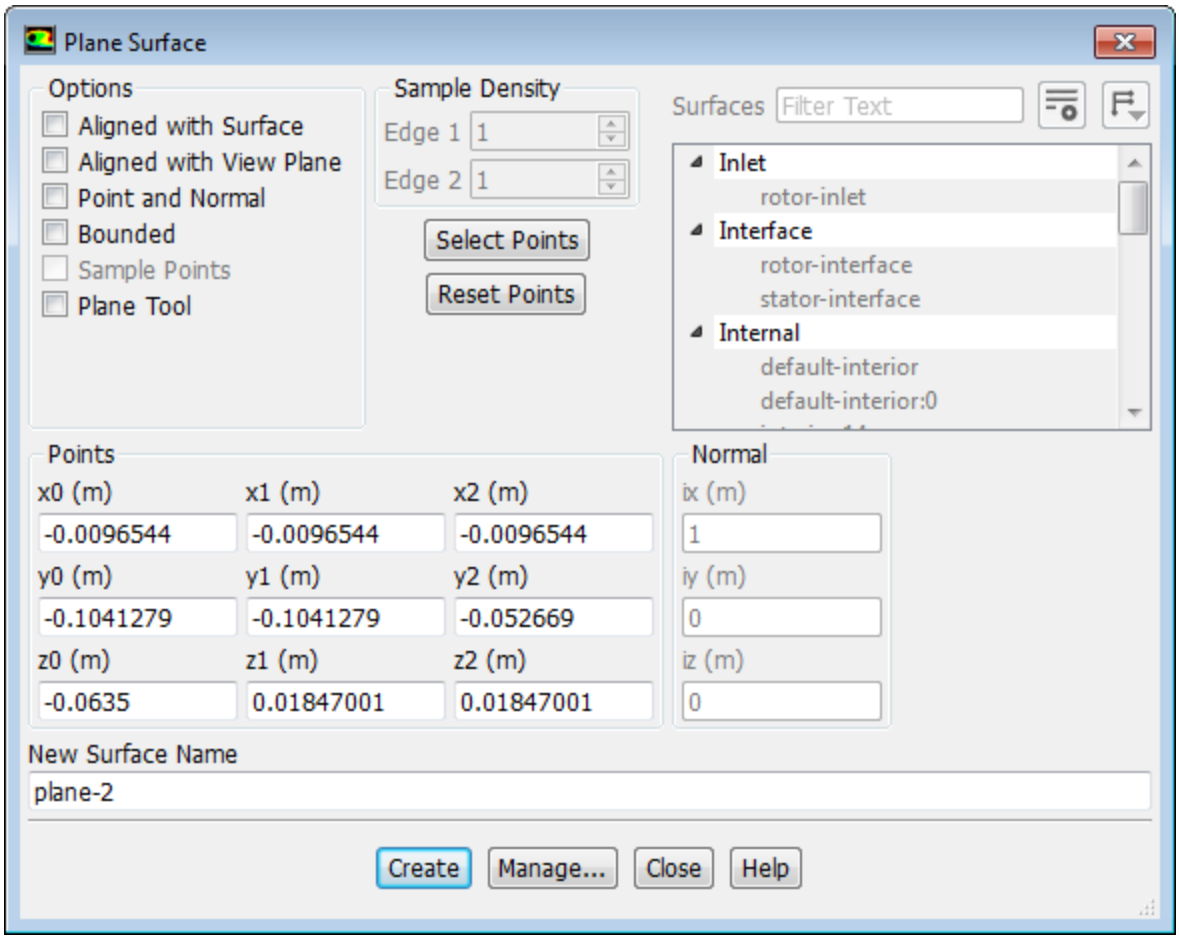
Height
specifies the height in pixels of the new window size.

Margin
specifies a margin in pixels to create around the picture. This can help you produce a video image that does not include unwanted parts of the screen, such as the window border.

38.2.68. Plane Surface Dialog Box

The **Plane Surface** dialog box allows you to interactively create a planar surface that cuts through the domain. See Plane Surfaces (p. 1722) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options
contains options for defining the plane surface.

Aligned with Surface
enables the specification of a plane parallel to an existing surface.

Aligned with View Plane
enables the specification of a plane parallel to the current view in the active graphics window.

Point and Normal
enables the specification of a plane having a certain normal vector and passing through a specified point.

Bounded
enables the creation of a bounded parallelepiped, 3 of whose 4 corners are the 3 points that define the plane equation (or the 4 corners of the **Plane Tool**).

Sample Points
enables the specification of a point density along the 2 directions (of the parallelepiped). This creates a uniformly distributed set of points on the plane. (This item is available only when the **Bounded** option is enabled.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plane Tool
enables the plane tool. See Using the Plane Tool (p. 1724) for details. This option is not available if you are using the **Aligned With Surface** or **Aligned With View Plane** option.

Sample Density
specifies the density of points when the **Sample Points** option is enabled.

Edge 1, Edge 2
sets the point density along the two directions of the plane. (Edge 1 extends from point 0 to point 1, and edge 2 extends from point 1 to point 2.)

Select Points
allows you to select points with the mouse. You can select endpoints by clicking on locations in the active window with the mouse-probe button. (See Controlling the Mouse Button Functions (p. 1800) for information about setting mouse button functions.)

Reset Points
resets the plane tool to its default position.

Surfaces
contains a list of currently defined surfaces. If you choose the **Aligned With Surface** option, this will become a selectable list, and you can choose the surface that you want the new plane surface to be aligned with.

Points
contains boxes in which you can set the coordinates of the three points defining the planar surface.

x0, y0, z0
designates the coordinates of the first point.

x1, y1, z1
designates the coordinates of the second point.

x2, y2, z2
designates the coordinates of the third point.

Normal
contains boxes in which you can specify the components of the normal vector when the **Point And Normal** option is enabled.

ix, iy, iz
designates the coordinates of the end point of the normal vector (the start point being 0, 0, 0).

New Surface Name
designates the name of the new surface. The default is the concatenation of the surface type and an integer which is the new surface ID.

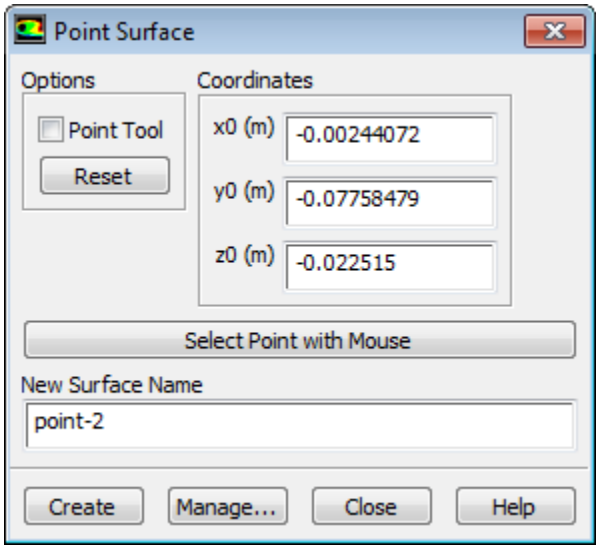
Create
creates the surface.

Manage...
opens the Surfaces Dialog Box (p. 2810) in which you can rename and delete surfaces and determine their sizes.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.2.69. Point Surface Dialog Box

The **Point Surface** dialog box allows you to interactively create point surfaces (surfaces containing a single data point). See Point Surfaces (p. 1716) for details about the items below.



Controls

Options
contains options related to the point tool. See Using the Point Tool (p. 1717) for details about using this feature.

Point Tool
enables the point tool.

Reset
resets the point tool to its default position.

Coordinates
designates the coordinates of the point in the surface (**x0**, **y0**, **z0**).

Select Point with Mouse
allows you to select the point with the mouse. You can select a point by clicking on a location in the active window with the mouse-probe button. (See Controlling the Mouse Button Functions (p. 1800) for information about setting mouse button functions.)

New Surface Name
designates the name of the new surface. The default is the concatenation of the surface type and an integer which is the new surface ID.

Create
creates the surface.

Manage...
opens the Surfaces Dialog Box (p. 2810) in which you can rename and delete surfaces and determine their sizes.

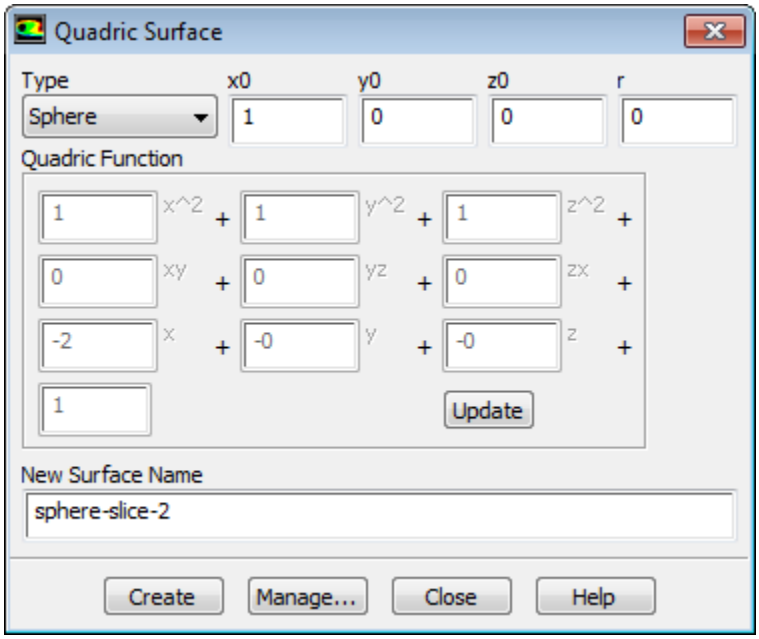
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.2.70. Quadric Surface Dialog Box

The **Quadric Surface** dialog box allows you to interactively define quadric functions and create surfaces from them. Lines and circles are commonly used types of quadric functions, so additional support is provided for defining them. See Quadric Surfaces (p. 1726) for details about using the items below.

Important

Note that your inputs for the quadric function must be in SI units.



Controls

Type
selects the type of quadric function. Presently, you may select **Line/Plane**, **Circle/Sphere** or a general **Quadric**. Some of the control buttons take on different meanings for each of the three quadric function types.

Line/Plane
specifies that the surface will consist of all points on the domain that satisfy the equation $ix*x+iy*y+iz*z=distance$, where

ix
is the coefficient of x in the quadric function.

iy
is the coefficient of y in the quadric function.

iz
is the coefficient of z in the quadric function.

distance
is the distance of the line/plane from the origin.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you click the **Update** button in **Quadric Function**, the coefficients in the quadric function will change to reflect your inputs.

Circle/Sphere
specifies that the surface will consist of all points in the domain that satisfy the equation $(x-x0)^2+(y-y0)^2+(z-z0)^2=r^2$, where

x0,y0,z0
are the x, y, z coordinates of the center.

r
is the radius.

When you click the **Update** button in **Quadric Function**, the coefficients in the quadric function will change to reflect your inputs.

Quadric
specifies that the surface will consist of all points in the domain that satisfy the quadric function $Q=value$, where Q is the quadric function you define as shown below. You will enter **value** to the right of the **Type** entry, but the coefficients in the quadric function are entered directly in the **Quadric Function** box. Note that in 2D problems the z entries are ignored.

x2, y2, z2, xy, yz, zx, x, y, z
are the coefficients of the corresponding terms in the quadric function.

Quadric Function
contains the display of the quadric function and the **Update** button. If you select **Line/Plane** or **Circle/Sphere** as the **Type**, the coefficients of the quadric function will be updated here when you click the **Update** button. You will not be able to edit the values in the **Quadric Function** box directly. If you select **Quadric** as the **Type**, you will enter the coefficients directly in this box, and the **Update** button will be inactive.

Update
updates the coefficients in **Quadric Function** to reflect the current input.

New Surface Name
designates the name of the new surface. The default is the concatenation of the surface type and an integer which is the new surface ID.

Create
creates the surface.

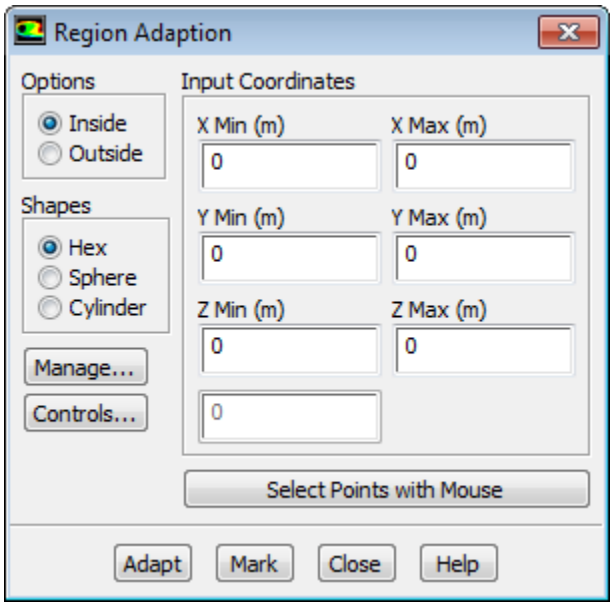
Manage...
opens the Surfaces Dialog Box (p. 2810) in which you can rename and delete surfaces and determine their sizes.

38.2.71. Region Adaption Dialog Box

The **Region Adaption** dialog box allows you to mark or refine cells inside or outside a specified region defined by text or mouse input. See Region Adaption (p. 1692) for details.

Setting Up Domain → Adapt → Mark/Adapt Cells → Region...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options
contains radio buttons that control whether the cells inside or outside the region are marked for refinement.

Inside
enables the marking of cells with centroids that are within the region.

Outside
enables the marking of cells with centroids that are outside the region.

Shapes
contains radio buttons that control the type of region.

Hex/Quad
defines a hexahedral region in 3D or a rectangular region in 2D. The appropriate button will appear for the solver you are using.

Sphere/Circle
defines a spherical region in 3D or a circular region in 2D. The appropriate button will appear for the solver you are using.

Cylinder
defines a cylindrical region in 3D or a rectangular region in 2D.

Manage...
opens the Manage Adaption Registers Dialog Box (p. 2728), which allows you to display and manipulate adaption registers that are generated using the **Mark** command.

Controls...
opens the Mesh Adaption Controls Dialog Box (p. 2730), which allows you to control certain aspects of the adaption process.

Input Coordinates
defines the extent of the selected region. The appearance of this box changes depending on the type of region selected.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the region selected is a hexahedron or quadrilateral, you will specify the minimum and maximum coordinates defining the box. (The **Radius** real number field will not be active.)

X Min, Y Min, Z Min
define the coordinates of the minimum point defining the hexahedron or rectangle. For quadrilaterals, the **Z Min** real entry field will not be active.

X Max, Y Max, Z Max
define the coordinates of the maximum point defining the hexahedron or rectangle. For quadrilaterals, the **Z Max** real entry field will not be active.

If the region selected is a sphere or circle, you will specify the coordinates of the sphere's center and its radius. (The maximum coordinate real number fields will not be active.)

X Center, Y Center, Z Center
are the coordinates of the centroid of the sphere or circle. For circles, the **Z Center** real entry field will not be active.

Radius
is the radius of the sphere or circle.

If the region selected is a cylinder, you will specify the minimum and maximum coordinates defining the cylinder axis, as well as the radius of the cylinder.

X-Axis Min, Y-Axis Min, Z-Axis Min
define the coordinates of the minimum point defining the cylinder axis. For 2D cases, the **Z-Axis Min** real entry field will not be active.

X-Axis Max, Y-Axis Max, Z-Axis Max
define the coordinates of the maximum point defining the cylinder axis. For 2D cases, the **Z-Axis Max** real entry field will not be active.

Radius
is the radius of the cylinder. (In 2D, this will be the width of the resulting rectangle.)

Select Points with Mouse
allows you to select specify coordinates with the mouse. If one of the mouse buttons is defined as a mouse probe, you may select the input coordinates from a display of the mesh or solution field. For more information on mouse buttons, refer to Controlling the Mouse Button Functions (p. 1800). After you select the points, the values will be loaded automatically into the appropriate real number field. You have the option of editing these values before marking or adapting. The order of input for defining a hexahedron (rectangle) is insignificant, but the order of input for the sphere (circle) has significance. First, you select the location of the centroid. Then you select a point that lies on the sphere, that is, a point that is one radius away from the centroid.

Adapt
adapts the mesh based on the region defined and the in/out option.

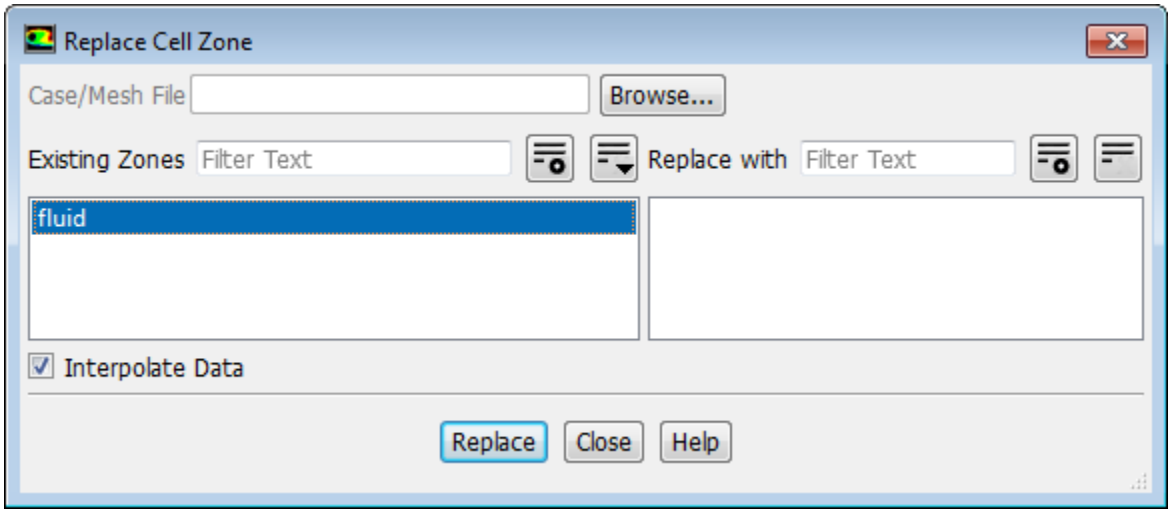
Mark
marks the cells to be refined based on the region defined and the in/out option. This command produces an adaption register. For information on using adaption registers, see Manipulating Adaption Registers (p. 1701).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.2.72. Replace Cell Zone Dialog Box

The **Replace Cell Zone** dialog box allows you to replace a single cell zone or multiple zones. See Replacing Zones (p. 218) for details.

Setting Up Domain → Zones → Replace Zone...



Controls

Case/Mesh File
allows you to specify a mesh file from which you want to replace the zone.

Existing Zones
contains a list of cell zones from which you can select the zone to be replaced.

Replace with
contains a list of cell zones from which you can select the zone to replace the zone selected in **Existing Zones** list.

Interpolate Data
allows you to enable/disable data interpolation if data already exists.

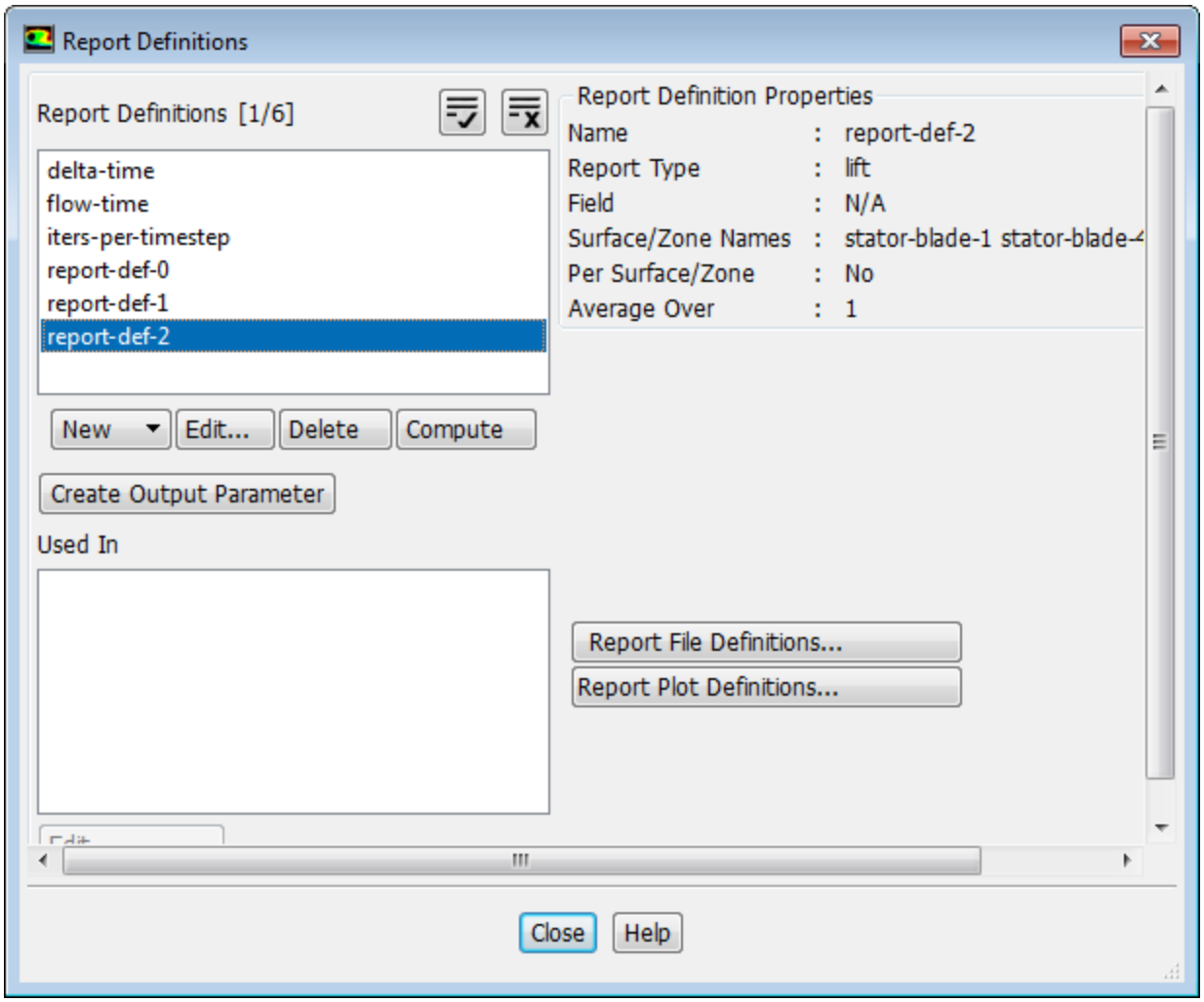
Replace
replaces the selected cell zone.

38.2.73. Report Definitions Dialog Box

Solving → Reports → Definitions → Edit...

For information on creating report definitions, see Monitoring and Reporting Solution Data (p. 1885).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Report Definitions
lists all of the existing report definitions.

New
drop-down list allowing you to create new report definitions (surface, volume, force, flux, dpm, and user-defined).

Edit...
allows you to edit the properties of the selected report definition.

Delete
deletes the selected report definition.

Compute
computes the value of the selected report definition and prints to the console.

Create Output Parameter
creates an output parameter for the selected report definition(s) with the name `<report definition name>-op`.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Used In
lists the report files and report plots that contain the selected report definition.

Edit...
opens the report file/plot dialog box for the report file/plot selected in the **Used In** list.

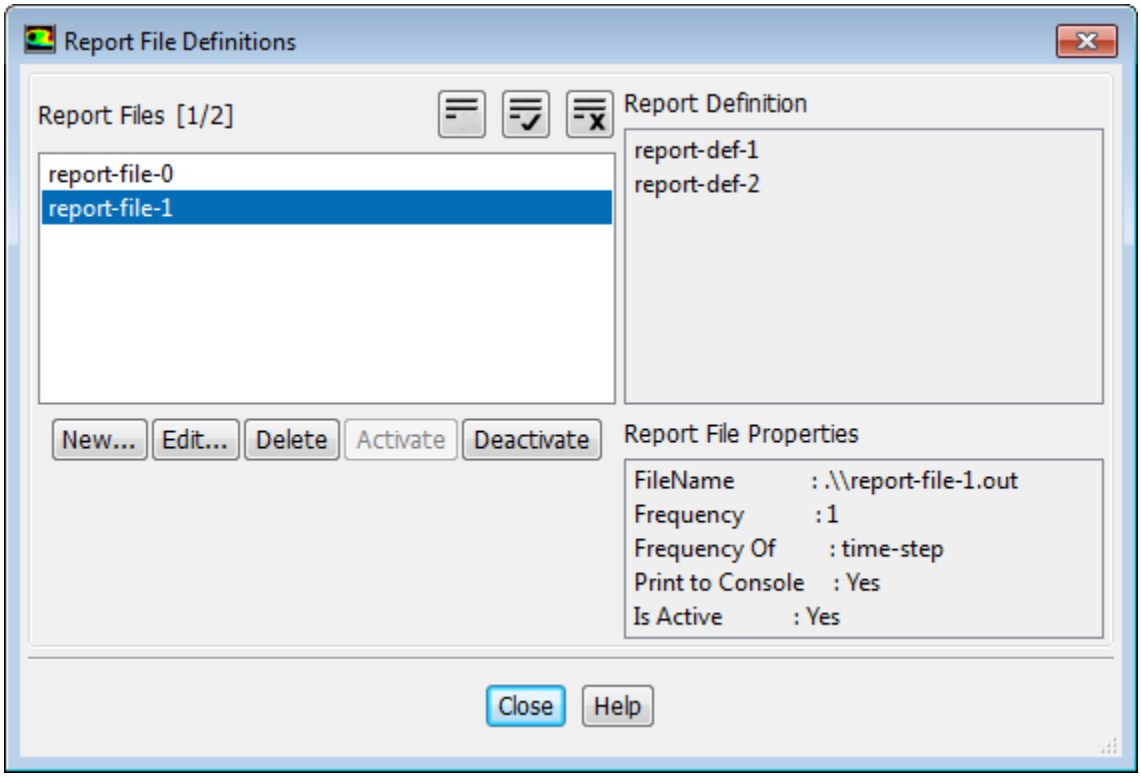
Report Definition Properties
displays the properties of the selected report definition in the **Report Definition** list.

Report File Definitions...
opens the Report File Definitions Dialog Box (p. 2787).

Report Plot Definitions
opens the Report Plot Definitions Dialog Box (p. 2788).

38.2.74. Report File Definitions Dialog Box

Solving → Reports → File...



Controls

Report Files
lists all of the existing report files.

New...
opens the New Report File Dialog Box (p. 2754).

Edit...
allows you to edit the properties of the selected report file in the Edit Report File Dialog Box (p. 2698).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Delete
deletes the selected report file.

Activate
activates the selected (previously deactivated) report file for writing during the solution calculation.

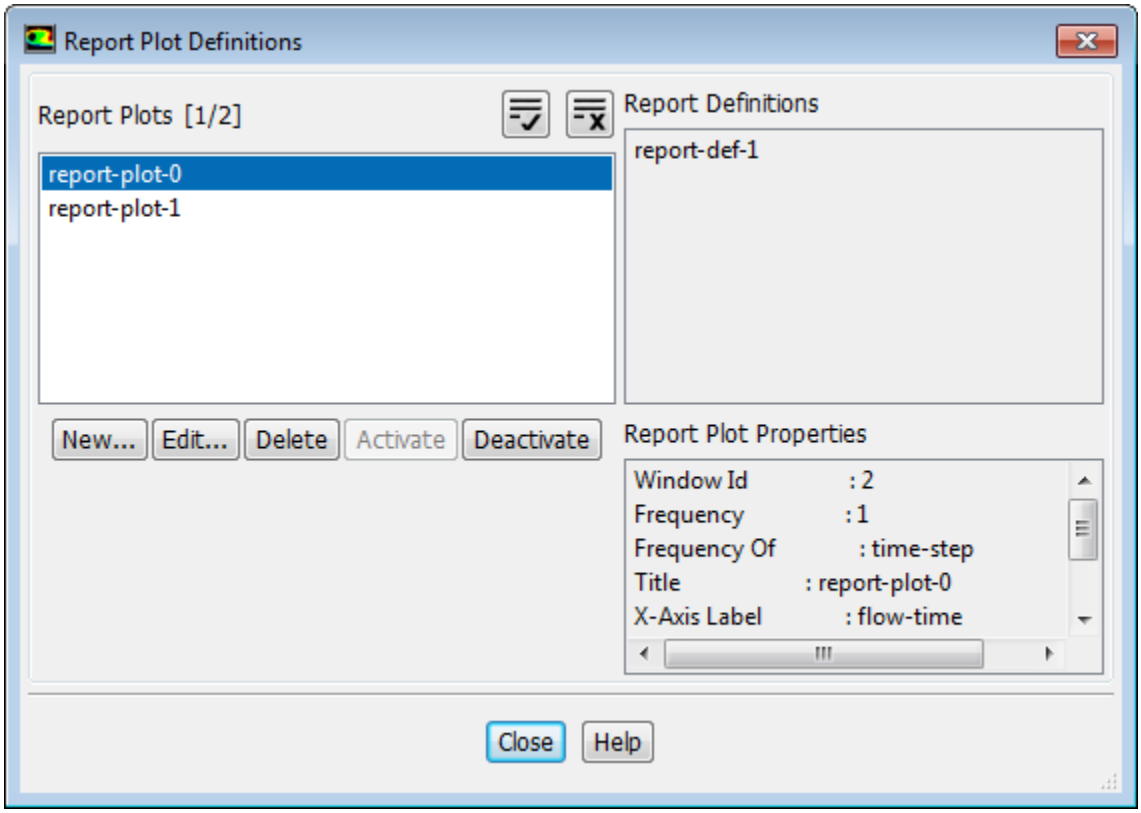
Deactivate
deactivates the writing of the selected report file during the solution calculation.

Report Definition
lists the report definitions included in the selected report file.

Report File Properties
lists the properties of the selected report file.

38.2.75. Report Plot Definitions Dialog Box

Solving → Reports → Plot...



Controls

Report Plots
lists all of the existing report plots.

New...
opens the New Report Plot Dialog Box (p. 2755).

Edit...
allows you to edit the properties of the selected report plot in the Edit Report Plot Dialog Box (p. 2700).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Delete
deletes the selected report plot.

Activate
activates the selected (previously deactivated) report plot for plotting during the solution calculation.

Deactivate
deactivates the plotting of the selected report plot during the solution calculation.

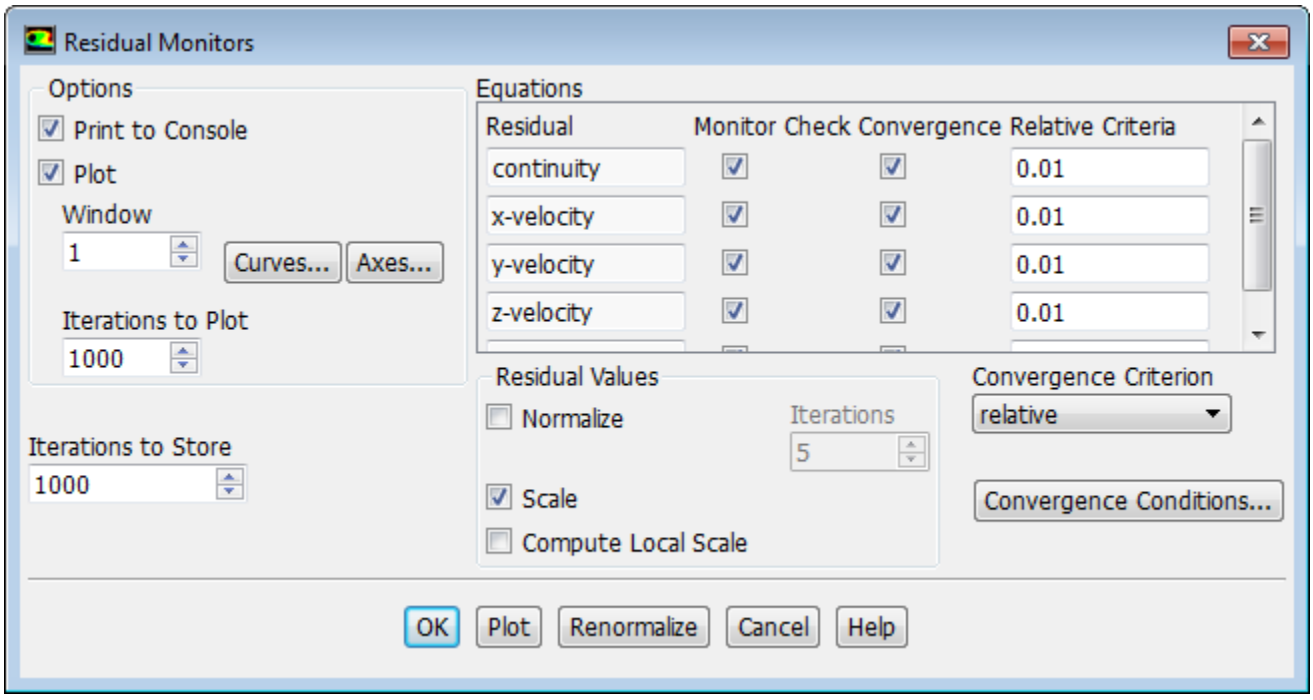
Report Definition
lists the report definitions included in the selected report plot.

Report Plot Properties
lists the properties of the selected report plot.

38.2.76. Residual Monitors Dialog Box

You can use the **Residual Monitors** dialog box to control the residual information that Fluent reports. See Monitoring Residuals (p. 1624) for details about the items below.

Solving → Reports → Residuals...



Controls

Options
selects any combination of the following methods for reporting residuals. See Printing and Plotting Residuals (p. 1627) for details.

Print to Console
specifies whether or not to print residuals in the text window after each iteration.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plot
specifies whether or not to plot residuals in the graphics window (with the window ID set in **Window**) after each iteration. See Plot Parameters (p. 1632) for details.

Window
sets the window ID in which the plot will be drawn. When ANSYS Fluent is iterating, the active graphics window is temporarily set to this window to update the residual plot, and then returned to its previous value. Thus, the residual plot can be maintained in a separate window that does not interfere with other graphical postprocessing.

Iterations to Plot
is the number of history points to display on the residual plot.

Axes...
opens the Axes Dialog Box (p. 2608), which allows you to modify the attributes of the axes.

Curves...
opens the Curves Dialog Box (p. 2610), which allows you to modify the attributes of the residual curves.

Iteration to Store
sets the number of residual history points to be stored in the data file. Due to the compaction algorithm used, saving 1000 points does not result in just the last 1000 iterations being saved; the history reaches back quite a bit further than that, but does not save a point at every iteration. Further back in the iteration history, the spacing between saved iterations grows larger. See Storing Residual History Points (p. 1628) for details.

Residual Values
controls the normalization and scaling of residuals. See Controlling Normalization and Scaling (p. 1628) for details.

Normalize
specifies whether or not to normalize the printed or plotted residual for each variable by the value indicated as the **Normalization Factor** for that variable. The default **Normalization Factor** is the maximum residual value after the first 5 iterations.

This option is off by default.

Iterations
sets the number of iterations for which ANSYS Fluent will search for the largest residual to normalize by. (If the **Normalize** option is turned off, this item will not be editable.)

Scale
specifies whether or not to print or plot scaled residuals for each variable. This option is on by default.

Compute Local Scale
computes and stores both the locally and globally scaled residuals from subsequent iterations, for the purpose of reporting. You will select the type of residual scaling form the **Reporting Option** drop-down list. See Definition of Residuals for the Pressure-Based Solver (p. 1624) and Definition of Residuals for the Density-Based Solver (p. 1626) for more information.

Reporting Option
gives you the choice of plotting or printing to the console the **global scaling** or **local scaling** of residuals.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Convergence Criterion
consists of four options that are available for checking an equation for convergence.

absolute
is the default and is available for steady-state cases. The residual (scaled and/or normalized) of an equation at an iteration is compared with a user-specified value. If the residual is less than the user-specified value, that equation is deemed to have converged for a time step.

relative
is where the residual of an equation at an iteration of a time step is compared with the residual at the start of the time step. If the ratio of the two residuals is less than a user-specified value, that equation is deemed to have converged for a time step.

relative or absolute
is where either the **absolute** convergence criterion or the **relative** convergence criterion is met. At that point, the equation is considered converged.

none
is used to disable convergence checking.

Convergence Conditions...
opens the Convergence Conditions Dialog Box (p. 2678).

Residual
indicates the name of each variable for which residual information is available.

Monitor
indicates whether or not the residuals for each variable are to be monitored. You can toggle monitoring on and off for each variable by turning the corresponding check box in the **Monitor** list on or off.

Normalization Factor
shows the normalization factor for each variable. The default is the maximum residual value after the first 5 iterations. To set this value manually, enter a new value in the corresponding **Normalization Factor** field. This list will not appear if the **Normalize** option is turned off.

Check Convergence
indicates whether or not the convergence of each variable is to be monitored. If convergence is being monitored, the solution will stop automatically when each variable meets its specified convergence criterion. You can check convergence only for variables for which you are monitoring residuals. You can toggle convergence checking on and off for each variable by turning the corresponding check box in the **Check Convergence** list on or off.

Absolute Criteria, Relative Criteria
shows the residual value for which the solution of each variable will be considered to be converged. To set this value manually, enter a new value in the corresponding **Absolute Criteria**/**Relative Criteria** field.

Plot
displays the current residual history plot.

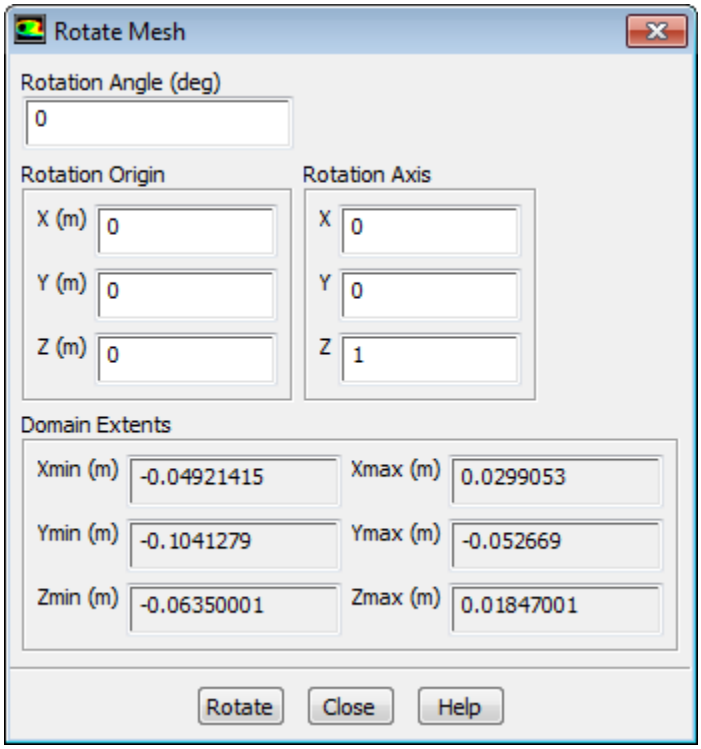
Renormalize
sets the normalization factors to the maximum values in the residual histories. **Renormalize** should be used to renormalize the residual plot in cases where the maximum residuals occur sometime after the first five iterations.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.2.77. Rotate Mesh Dialog Box

The **Rotate Mesh** dialog box allows you to rotate the mesh about the required axis and rotation origin by specifying the angle of rotation. See Rotating the Mesh (p. 229) for details.

Setting Up Domain → Mesh → Transform → Rotate...



Controls

Rotation Angle
is the angle with which you want to rotate the mesh. You can enter a positive or negative real number.

Rotation Origin
defines the new origin for the mesh rotation.

Rotation Axis
defines the axis about which you want to rotate the mesh.

Domain Extents
displays the Cartesian coordinate extremes of the nodes in the mesh. (These values are not editable; they are purely informational.)

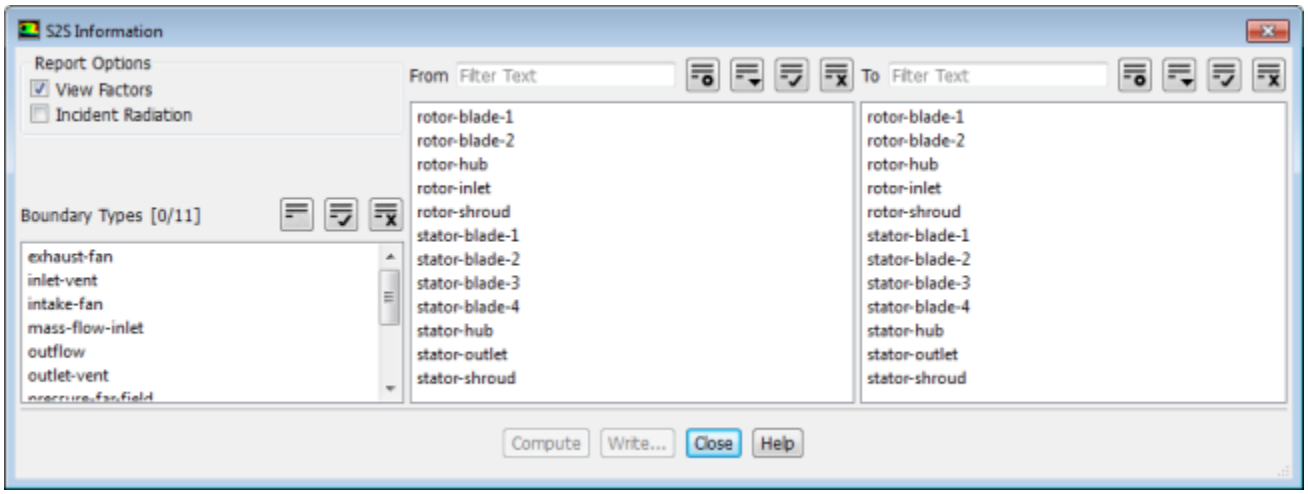
Rotate
adds the specified rotation parameters to the appropriate Cartesian coordinate of every node in the mesh.

38.2.78. S2S Information Dialog Box

The **S2S Information** dialog box allows you to report values of the view factor and radiation emitted from one zone to any other zone. See Reporting Radiation in the S2S Model (p. 885) for details about the items below.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Postprocessing → Model Specific → S2S Information...



Controls

Report Options
contains items for which information is available for reporting.

View Factors
turns on the computation of view factors from one zone to the other.

Incident Radiation
turns on the computation of the incident radiation from one zone to the other.

Boundary Types
contains a selectable list of types of boundary zones. If you select (or deselect) an item in this list, all zones of that type will be selected (or deselected) automatically in the **From** and **To** lists.

From
contains a selectable list of boundary zones for which you would like data reported from the selected zone.

To
contains a selectable list of boundary zones for which you would like data reported to the selected zone.

Compute
computes the view factors and/or incident radiation on the selected zones.

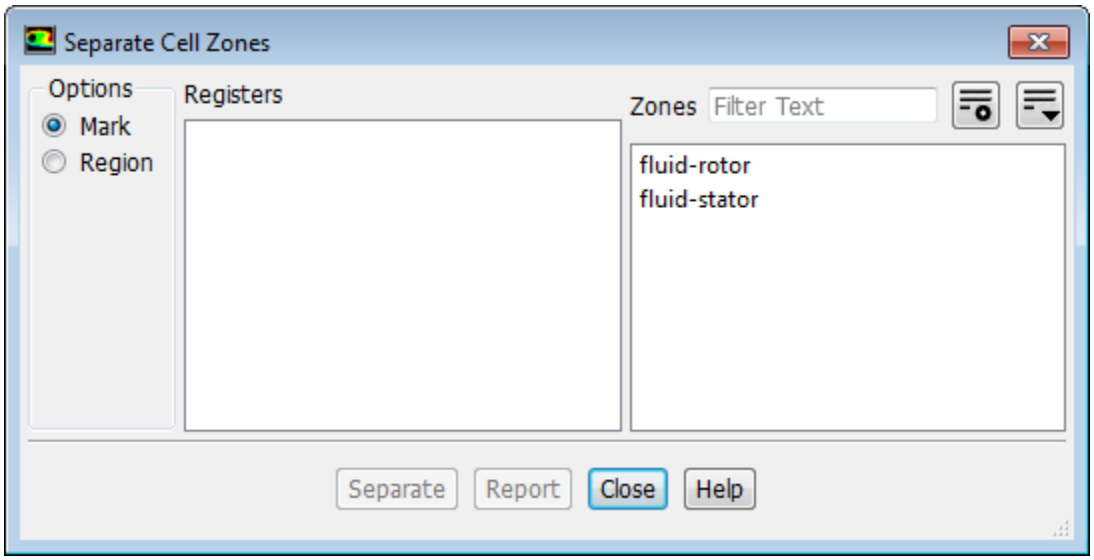
Write...
opens The Select File Dialog Box (p. 19), which you can use to save the data as an S2S Info File (`.sif` format).

38.2.79. Separate Cell Zones Dialog Box

The **Separate Cell Zones** dialog box allows you to separate a single cell zone into multiple zones of the same type. See Separating Zones (p. 208) for details.

Setting Up Domain → Zones → Separate → Cells...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options
specifies the method on which the cell-zone separation is to be based.

Mark
indicates that the cell zone is to be separated based on the marks stored in adaption registers.

Region
indicates that the cell zone is to be separated into two or more contiguous regions based on an internal boundary within the original zone.

Registers
contains a list of defined adaption registers. If you are separating the cell zone by mark, select the adaption register to be used in the **Registers** list. When the separation is performed, cells that are marked will be placed into a new zone.

Zones
contains a list of cell zones from which you can select the zone to be separated.

Separate
separates the selected cell zone based on the specified parameters.

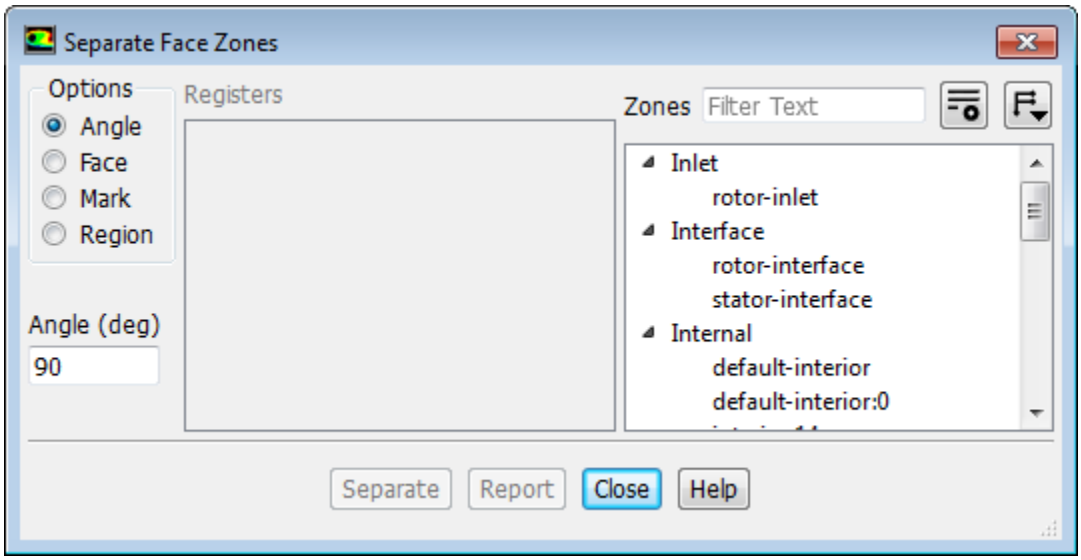
Report
reports what the result of the separation will be without actually separating the cell zone.

38.2.80. Separate Face Zones Dialog Box

The **Separate Face Zones** dialog box allows you to separate a single face zone into multiple zones of the same type. See Separating Zones (p. 208) for details.

Setting Up Domain → Zones → Separate → Faces...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options
specifies the method on which the face separation is to be based.

Angle
indicates that the face zone is to be separated based on significant angle (specified in the **Angle** field).

Face
indicates that the face zone is to be separated by putting each face in the zone into its own zone.

Mark
indicates that the face zone is to be separated based on the marks stored in adaption registers.

Region
indicates that the face zone is to be separated based on contiguous regions.

Registers
contains a list of defined adaption registers. If you are separating faces by mark, select the adaption register to be used in the **Registers** list. When the separation is performed, all faces of cells that are marked will be placed into a new face zone.

Zones
contains a list of face zones from which you can select the zone to be separated.

Angle
specifies the significant angle to be used when you separate a face zone based on angle. Faces with normal vectors that differ by an angle greater than or equal to the specified significant angle will be placed in different zones when the separation occurs.

Separate
separates the selected face zone based on the specified parameters.

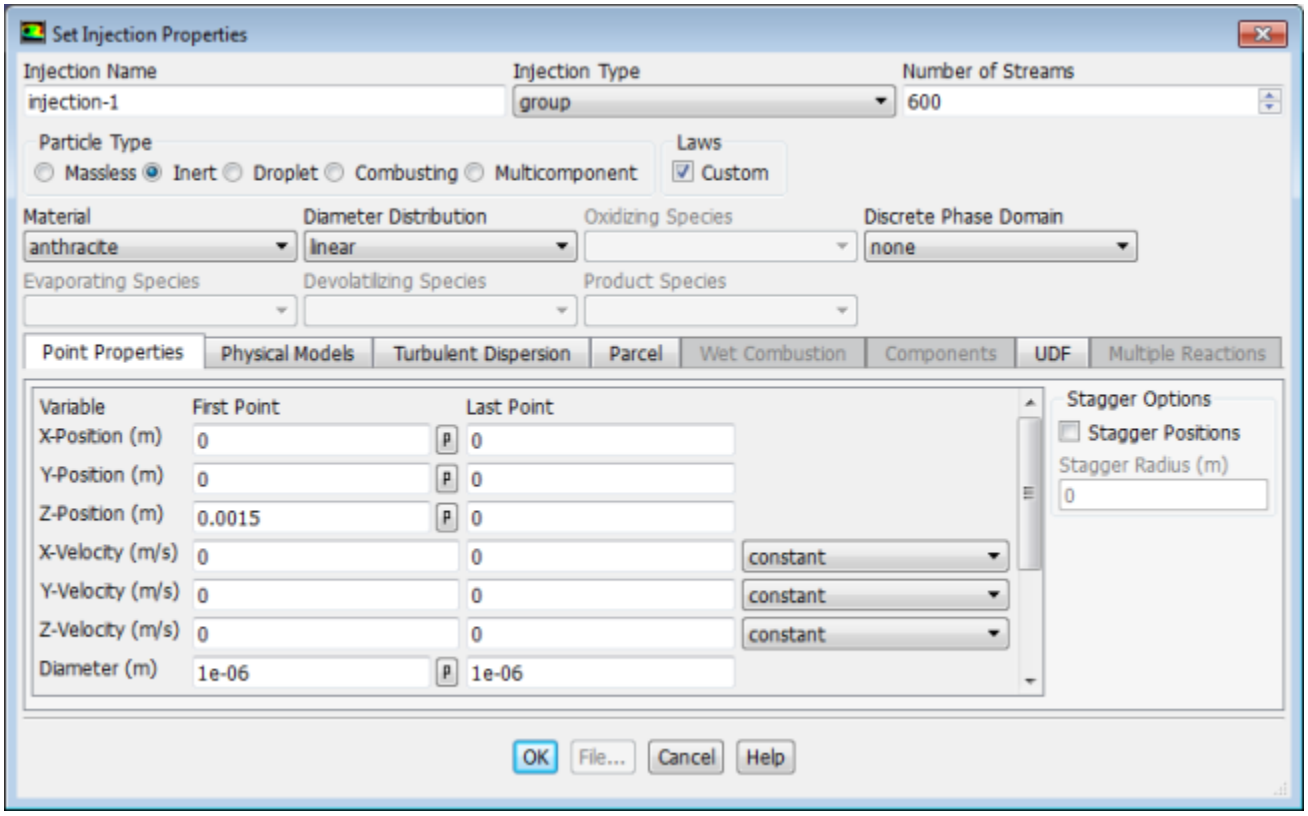
Report
reports what the result of the separation will be without actually separating the face zone.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.2.81. Set Injection Properties Dialog Box

The **Set Injection Properties** dialog box allows you to define the properties of an existing discrete-phase injection (which was created in the Injections Dialog Box (p. 2716)). See Defining Injection Properties (p. 1286) for details about the items listed in this section.

The **Set Injection Properties** dialog box is opened from the Injections Dialog Box (p. 2716).



Controls

Injection Name

sets the name of the injection.

Injection Type

contains a drop-down list of the available injection types: **single**, **group**, **cone**, **surface**, **plain-orifice-atomizer**, **pressure-swirl-atomizer**, **air-blast-atomizer**, **flat-fan-atomizer**, **effervescent-atomizer**, and **file**. (**cone** is not available in 2D.) These choices are described in Injection Types (p. 1266).

Number of Streams

indicates the number of particle streams in a **group**, **cone**, or any of the **atomizer** injections. (This item will not appear for **single**, **surface**, or **file** injections.)

Release From Surfaces

indicates the surface from which the particles in a **surface** injection will be released. (This item will appear only for a **surface** injection.)

Particle Type

specifies the particle type as **Massless**, **Inert**, **Droplet**, **Combusting**, or **Multicomponent**. These types are described in Particle Types (p. 1268).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Laws
(not for massless particles) contains inputs for customized particle laws.

Custom
enables the specification of customized particle laws and opens the Custom Laws Dialog Box (p. 2684).

Material
(not for massless particles) indicates the material for the particles. If this is the first time you have created a particle of this type, you can choose from all of the materials by copying them from the database or creating them from scratch, as discussed in Setting Discrete-Phase Physical Properties (p. 1316) and described in detail in Using the **Create/Edit Materials** Dialog Box (p. 461).

Diameter Distribution
(not for massless particles) allows you to change from the default **linear** interpolation method used to determine the size of the particles in a **group** injection, or the default **uniform** method used to determine the size of the particles in a **surface** injection, to the **rosin-rammler** or **rosin-rammler-logarithmic** method. The Rosin-Rammler method for determining the range of diameters is described in Using the Rosin-Rammler Diameter Distribution Method (p. 1281).

Evaporating Species
(for **droplet** particles) specifies the gas-phase species created by the vaporization and boiling laws (laws 2 and 3).

Devolatilizing Species
(for **combusting** particles) specifies the gas-phase species created by the devolatilization law (law 4).

This item will not appear for two-mixture-fraction non-premixed combustion calculations.

Devolatilizing Stream
(for **combusting** particles) specifies the destination stream for the gas-phase species created by the devolatilization law (law 4).

This item will appear only for two-mixture-fraction non-premixed combustion calculations.

Oxidizing Species
(for **combusting** particles) specifies the gas phase species that participates in the surface char combustion reaction (law 5).

Product Species
(for **combusting** particles) specifies the gas-phase species created by the surface char combustion reaction (law 5).

This item will not appear for two-mixture-fraction non-premixed combustion calculations.

Product Stream
(for **combusting** particles) specifies the destination stream for the gas-phase species created by the surface char combustion reaction (law 5).

This item will appear only for two-mixture-fraction non-premixed combustion calculations.

Discrete Phase Domain
is available when using the **Dense Discrete Phase Model**, described in Including the Dense Discrete Phase Model (p. 1481).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DEM Collision Partner
is available when using the **DEM Collision** model, described in Modeling Collision Using the DEM Model (p. 1253).

Point Properties
displays the inputs for the point properties for the injection (for example, position, velocity, diameter, temperature, and mass flow rate). These inputs are described for each injection type in Point Properties for Single Injections (p. 1269) – Point Properties for Effervescent Atomizer Injections (p. 1280).

First Point
specifies the first point properties for the injection.

Last Point
specifies the last point properties for the injection.

Physical Models
displays the inputs for injection-specific physics models.

Drag Parameters
allows the setting of the drag law used in calculating the force balance on the particles. See Particle Force Balance in the Theory Guide for details on the items below.

Drag Law
is a drop-down list containing the available choices for the drag law:

spherical
assumes that the particles are smooth spheres.

nonspherical
assumes that the particles are not spheres, but are all identically shaped. The shape is specified by the **Shape Factor**.

Stokes-Cunningham
is for use with sub-micron particles. A **Cunningham Correction** is added to Stokes' drag law to determine the drag.

high-Mach-number
is similar to the spherical law with corrections to account for a particle Mach number greater than 0.4 or a particle Reynolds number greater than 20.

Ishii-Zuber
is used in gas-liquid flows. This drag law is only available if the magnitude and direction of the gravity vector are specified in the **Operating Conditions** dialog box. See Ishii-Zuber Drag Model in the *Fluent Theory Guide* for details.

Grace
is used in gas-liquid flows. This drag law is only available if the magnitude and direction of the gravity vector are specified in the **Operating Conditions** dialog box. See Grace Drag Model in the *Fluent Theory Guide* for details.

dynamic-drag
accounts for the effects of droplet distortion. This drag law is available only when one of the droplet breakup models is used in conjunction with unsteady tracking. See Dynamic Drag Model Theory in the Theory Guide for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Shape Factor
specifies the shape of the particles when **nonspherical** is selected as the **Drag Law** (φ in Equation 16.75 in the Theory Guide). It is the ratio of the surface area of a sphere having the same volume as the particle to the actual surface area of the particle. The shape factor value cannot be greater than 1.

Cunningham Correction
(C_c in Equation 16.77 in the Theory Guide) is used with Stokes' drag law to determine the force acting on the particles when the particles are sub-micron size. It appears when **Stokes-Cunningham** is selected as the **Drag Law**.

Particle Rotation
contains settings for the modeling of particle rotation when the **Enable Rotation** check box is selected (see Wall-Particle Reflection Model Theory in the *Fluent Theory Guide*).

Rotational Drag Law
is a drop-down list of available rotational drag law models:

Dennis-et-al
uses Equation 16.4 in the *Fluent Theory Guide* to calculate the rotational drag coefficient C_ω.

none
implies zero torque, that is, it excludes the influence of the fluid on the rotational particle motion.

Magnus Lift Law
is a drop-down list of available Magnus lift law models:

Oesterle-Bui-Dinh
uses Oesterle and Bui Dinh's formulation (Equation 16.18 in the *Fluent Theory Guide*).

Tsuji-et-al
is based on the particle spin rate (Equation 16.19 in the *Fluent Theory Guide*).

Rubinow-Keller
uses a linear dependence of the rotational lift coefficient C_{RL} on the spin rate (Equation 16.21 in the *Fluent Theory Guide*).

none
implies that the Magnus lift will not be included in your simulation.

Brownian Motion
enables the incorporation of the effects of Brownian motion. This option is available only when the **Stokes-Cunningham** drag law is selected from the **Drag Law** drop-down list. See Brownian Force in the Theory Guide for details.

Breakup
contains parameters that control droplet breakup and collision. (This section of the dialog box appears only if **Unsteady Tracking** is enabled in the Discrete Phase Model Dialog Box (p. 2278).)

Enable Breakup
enables breakup for this injection.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Breakup Model

is a drop-down list containing parameters that control droplet breakup. (This item appears only if **Enable Breakup** is selected.)

TAB

enables the Taylor Analogy Breakup (TAB) model, which is applicable to many engineering sprays. This method is based upon Taylor's analogy between an oscillating and distorting droplet and a spring mass system. See Taylor Analogy Breakup (TAB) Model in the Theory Guide for details.

For the **TAB** model, you must specify the following constants:

y0

is the constant y_0 in Equation 16.370 in the Theory Guide.

Breakup Parcels

is the number of child parcels the droplet is split into, as described in Velocity of Child Droplets in the Theory Guide.

Wave

enables the Wave breakup model, which considers the breakup of the injected liquid to be induced by the relative velocity between the gas and liquid phases. See Wave Breakup Model for details.

For the **Wave** model, you must specify the following constants:

B0

is the constant B_0 in Equation 16.397 in the Theory Guide.

B1

is the constant B_1 in Equation 16.399 in the Theory Guide.

KHRT

enables the Kelvin-Helmholtz Rayleigh-Taylor breakup model, which considers the two competing effects of aerodynamic breakup and instabilities due to droplet acceleration. See KHRT Breakup Model in the Theory Guide for details.

For the **KHRT** model, you must specify the following constants:

B0

is the constant B_0 in Equation 16.397 in the Theory Guide.

B1

is the constant B_1 in Equation 16.399 in the Theory Guide.

Ctau

is the constant C_τ in Equation 16.404 in the Theory Guide.

CRT

is the constant C_{RT} in Equation 16.405 in the Theory Guide.

CL

is the constant C_L in Equation 16.400 in the Theory Guide.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SSD

enables the stochastic secondary droplet breakup model, where the probability of breakup is independent of the parent droplet size and the secondary droplet size is sampled from an analytical solution of the Fokker-Planck equation for the probability distribution. See Stochastic Secondary Droplet (SSD) Model in the Theory Guide for details.

For the **SSD** model, you must specify the following constants:

Critical We

is the critical Weber number in Equation 16.406 in the Theory Guide.

Core B1

is B in Equation 16.407 in the Theory Guide.

Target Np

is the average number in parcels for daughter parcels.

Xi

is $\langle \xi \rangle$ in Equation 16.408 in the Theory Guide.

Turbulent Dispersion

displays the inputs for stochastic tracking and cloud tracking.

Stochastic Tracking

controls the stochastic tracking for turbulent flows. Stochastic tracking includes the effect of turbulent velocity fluctuations on the particle trajectories using the DRW model described in Stochastic Tracking in the Theory Guide. See Stochastic Tracking (p. 1294) for details about the items below.

Discrete Random Walk Model

includes the effect of instantaneous turbulent velocity formulations on the particle trajectories through stochastic method.

Random Eddy Lifetime

specifies that the characteristic lifetime of the eddy is to be random.

Number of Tries

controls the inclusion of turbulent velocity fluctuations.

An input of 1 or greater tells ANSYS Fluent to include turbulent velocity fluctuations in the particle force balance.

Time Scale Constant

is C_L in Equation 16.24 in the Theory Guide. The default is 0.15; if you use the RSM, a value of 0.3 is recommended.

Cloud Tracking

incorporates the effects of turbulent dispersion on the injection. For details on the following items, see Particle Cloud Tracking in the Theory Guide and Cloud Tracking (p. 1296).

Cloud Model

enables particle cloud tracking.

Min. Cloud Diameter

specifies the diameter of the cloud in which the particles enter the domain.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Max. Cloud Diameter
specifies the maximum allowed cloud diameter.

Parcel
displays the inputs for controlling the discrete phase parcels.

Parcel Definitions
contains settings for how parcels are defined

Parcel Release Method
specifies the method for releasing parcels. Available settings are **standard**, **constant-number**, **constant-mass**, and **constant-diameter**. See Steady/Transient Treatment of Particles (p. 1242) for details about these settings.

Wet Combustion
displays the inputs for the wet combustion model.

Wet Combustion Model
allows the combusting particles to include an evaporating/boiling material.

Liquid Material
contains a drop-down list of liquid materials that can be chosen as the evaporating/boiling material to be included with the combusting particles.

Liquid Fraction
sets the volume fraction of the liquid present in the particle.

Components
displays the inputs for **Multicomponent** for use in the definition of the particle injection. For details on the following items, see Vapor Liquid Equilibrium Theory in the Theory Guide.

Multicomponent Settings
contains the multicomponent injections.

Component
specifies the component that is a part of the multicomponent species.

Mass Fraction
specifies the mass fraction of the component in a multicomponent species.

Evaporating Species
specifies the gas-phase species to be evaporated.

Evaporating Stream
specifies the source stream from which the species will be evaporated.

UDF
displays the inputs for **User-Defined Functions** for use in the definition of the particle injection. For details about user-defined functions, see the separate Fluent Customization Manual.

Initialization
contains a drop-down list of available user-defined functions. The UDF that you choose will be used to modify the injection properties at the time the particles are injected into the domain.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Heat/Mass Transfer
allows you to select the user-defined function that defines the heat or mass transfer.

Multiple Reactions
displays the inputs for **Multiple Surface Reactions**. See Particle Surface Reactions (p. 1001) for details about this model.

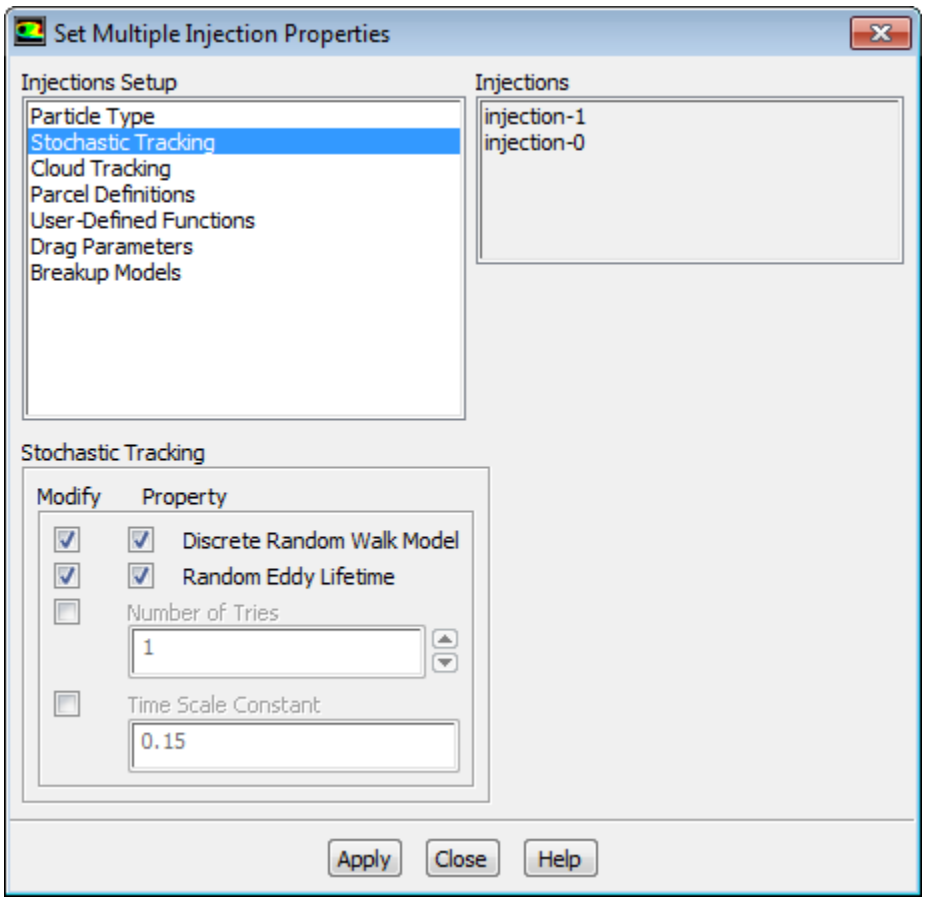
Species Mass Fractions
specify the combustible fraction of the combusting particle if you have defined more than one particle surface species. See Using the Multiple Surface Reactions Model for Discrete-Phase Particle Combustion (p. 1002) for details.

File...
opens The Select File Dialog Box (p. 19), in which you can select a file containing the injection definition (when **file** is selected as the **Injection Type**).

38.2.82. Set Multiple Injection Properties Dialog Box

The **Set Multiple Injection Properties** dialog box allows you to set properties that are common to multiple injections. See Defining Properties Common to More than One Injection (p. 1297) for details about the items below.

The **Set Multiple Injection Properties** dialog box is opened from the Injections Dialog Box (p. 2716).



Controls

Injections Setup
contains a list of the categories of injection properties that you can set for the injections in the **Injections** list. These categories correspond to the categories of inputs in the Set Injection Properties Dialog Box (p. 2796). When you select an item in the **Injections Setup** list, the dialog box will expand to show the relevant inputs, which are the same as those in the **Set Injection Properties** dialog box.

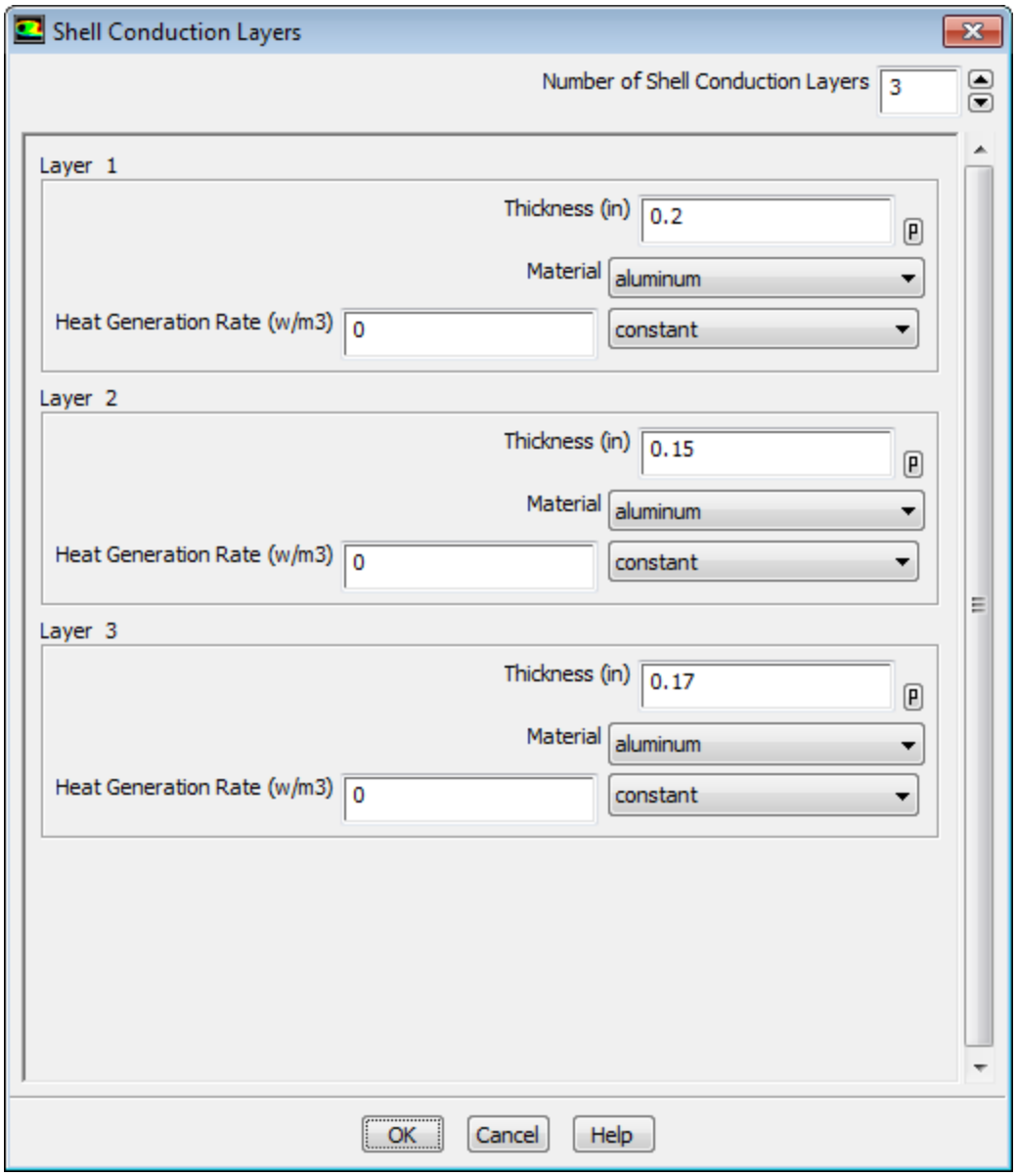
Injections
displays an informational list of the injections for which you are setting common properties. These are the injections that you selected in the Injections Dialog Box (p. 2716).

38.2.83. Shell Conduction Layers Dialog Box

The **Shell Conduction Layers** dialog box allows you to define the shell conduction settings for either a single **Wall** dialog box or all of the walls selected in the **Shell Conduction Zones** list of the Shell Conduction Manager Dialog Box (p. 2806). See Shell Conduction (p. 376) and Managing Shell Conduction Walls (p. 840) for details about the items below.

The **Shell Conduction Layers** dialog box is opened from either the Wall Dialog Box (p. 2456) or the Shell Conduction Manager Dialog Box (p. 2806).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Number of Shell Conduction Layers

allows you to define the number of layers that make up the wall(s).

Layer 1, Layer 2...

allows you to define the settings for each layer. Note that **Layer 1** refers to the layer closest to the fluid / solid adjacent to the wall zone, and layers with higher numbers are further away.

Thickness

sets the thickness of the layer for the calculation of the shell thermal resistance. Each layer must have a nonzero thickness

Material Name

sets the material type for the layer. The conductivity of the material is used for the calculation of shell thermal resistance. Materials are defined using the Materials Task Page (p. 2300).

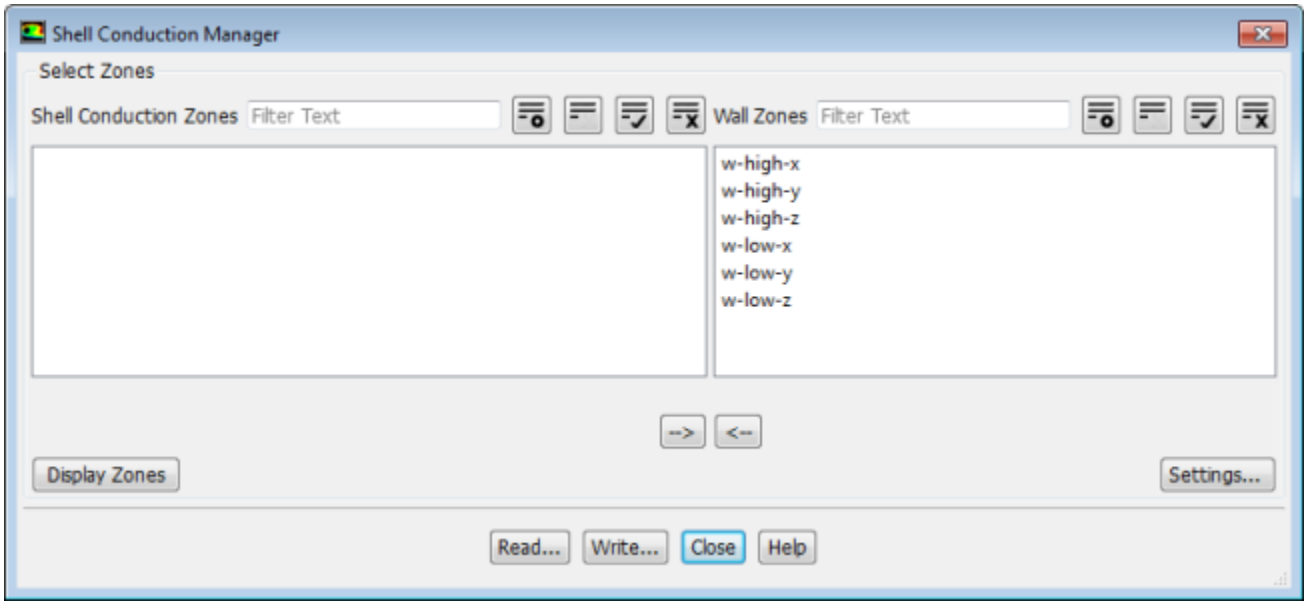
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Heat Generation Rate
sets the rate of heat generation in the layer. Note that you have the option of defining the heat generation rate using a user-defined function (UDF) that utilizes the `DEFINE_PROFILE` macro; for more information on creating and using user-defined functions, see the Fluent Customization Manual.

38.2.84. Shell Conduction Manager Dialog Box

The **Shell Conduction Manager** dialog box allows you to manage, define, and display shell conduction zones all in one location. See Managing Shell Conduction Walls (p. 840) for details about using the **Shell Conduction Manager** dialog box.

Setting Up Physics → Model Specific → Shell Conduction...



Controls

Select Zones
contains the list of **Shell Conduction Zones** and **Wall Zones**.

Shell Conduction Zones
contains a list of zones with shell conduction enabled.

Wall Zones
contains a list of zones without shell conduction.

-> , <-
disables and enables shell conduction for a selected zone, respectively.

Display Zones
displays the selected walls in the graphics window. Note that you can select walls with or without shell conduction. The zones will be displayed with different colors depending on the option selected in Mesh Colors Dialog Box (p. 2168) (accessible from the Mesh Display Dialog Box (p. 2163)).

Settings...
opens the Shell Conduction Layers Dialog Box (p. 2804), where you can define the shell properties of the walls selected in the **Shell Conduction Zones** list.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Read...
allows you to define your shell conduction settings by reading a CSV file.

Write...
allows you to write your saved shell conduction settings to a CSV file.

38.2.85. Surface Meshes Dialog Box

The **Surface Meshes** dialog box allows you to read the surface meshes in ANSYS Fluent (See Reading Surface Mesh Files (p. 149) for details on reading surface meshes.)

The **Surface Meshes** dialog box is opened from the Geometry Based Adaption Dialog Box (p. 2709).

Controls

Surfaces
contains a list of the surfaces available in the surface mesh you read.

*You can select/deselect the surfaces listed under **Surfaces**.*

Read...
opens the **Select File** dialog box in which you can select the surface mesh you want to read.

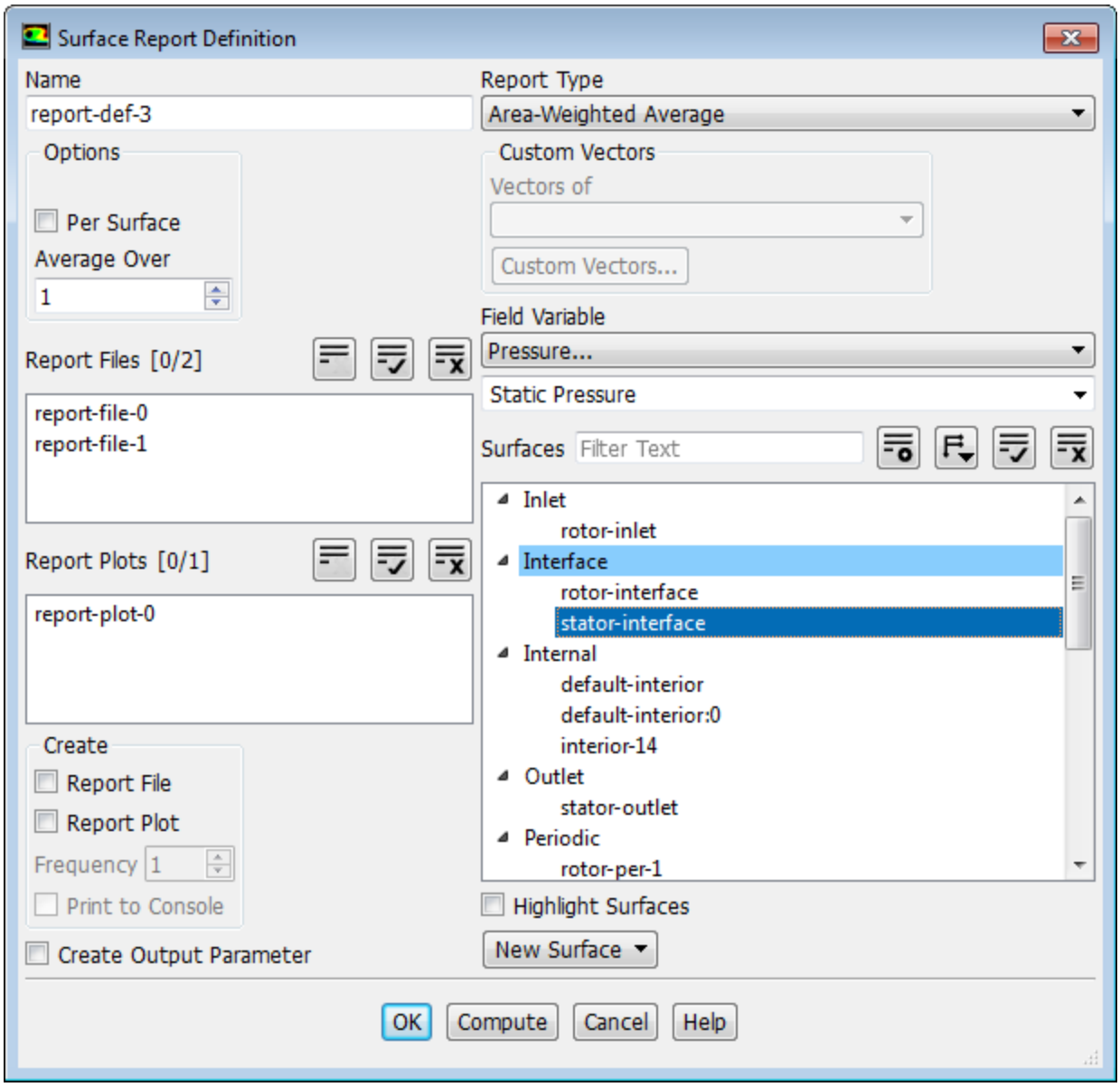
Delete
allows you to delete the selected surfaces under **Surfaces** area.

Display
allows you to display the selected surfaces under **Surfaces**.

38.2.86. Surface Report Definition Dialog Box

Solving → Reports → Definitions → New → Surface Report

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options

Per Surface
specifies whether or not the chosen field variable is calculated from all of the selected surfaces combined (default) or individually on each of the selected surfaces.

Average Over
(optional) To have Fluent calculate a running average for the **Surface Report Definition** you can enter a positive integer greater than 1 (the default) for **Average Over**.

Specifying a number greater than 1 means that Fluent will print, plot, and write the running average value of the selected variable instead of the current value of the same variable.

The value reported is averaged over the last N *iterations/time steps, where* N *is your specified* ***Average Over*** *value. When the iteration/time step number is lower than* N*, Fluent calculates the average of the available variable values.*

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Report Files
lists all of the report files where you can write the report definition data.

Report Plots
lists all of the report plots where you can plot the report definition data (assuming this report has the same units as the other report definitions included in the selected report plot).

Create

Report File
creates a new report file that includes this report definition.

Report Plot
creates a new report plot that includes this report definition.

Frequency
specifies how frequently the report definition is written, plotted, or printed to the console.

Print to Console
prints the value of the report definition to the console.

Create Output Parameter
creates an output parameter for this report definition with the name `<report definition name>-op`.

Report Type
selects the integration method used on the selected surfaces. The available report types are the same as those in the Surface Integrals Dialog Box (p. 2616). See Surface Integration (p. 1920) for details.

Custom Vectors
allows you to specify a vector if you selected **Custom Vector Based Flux**, **Custom Vector Flux** or **Custom Vector Weighted Average** from the **Report Type** drop-down list.

Field Variable
contains a list of solution variables that can be monitored on the selected surfaces. This list will not be available if you select **Mass Flow Rate** or **Volume Flow Rate** as the **Report Type**.

Phase
contains a list of all of the phases in the problem that you have defined. This is available when the VOF, mixture, or Eulerian multiphase model is enabled.

Surfaces
contains a selectable list of the current surfaces.

Highlight Surfaces
if enabled, this option displays the selected surface(s) in the graphics window.

New Surface
is a drop-down list button that contains a list of surface options:

Point
opens the Point Surface Dialog Box (p. 2780).

Line/Rake
opens the Line/Rake Surface Dialog Box (p. 2727).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plane
opens the Plane Surface Dialog Box (p. 2777).

Quadric
opens the Quadric Surface Dialog Box (p. 2781).

Iso-Surface
opens the Iso-Surface Dialog Box (p. 2721).

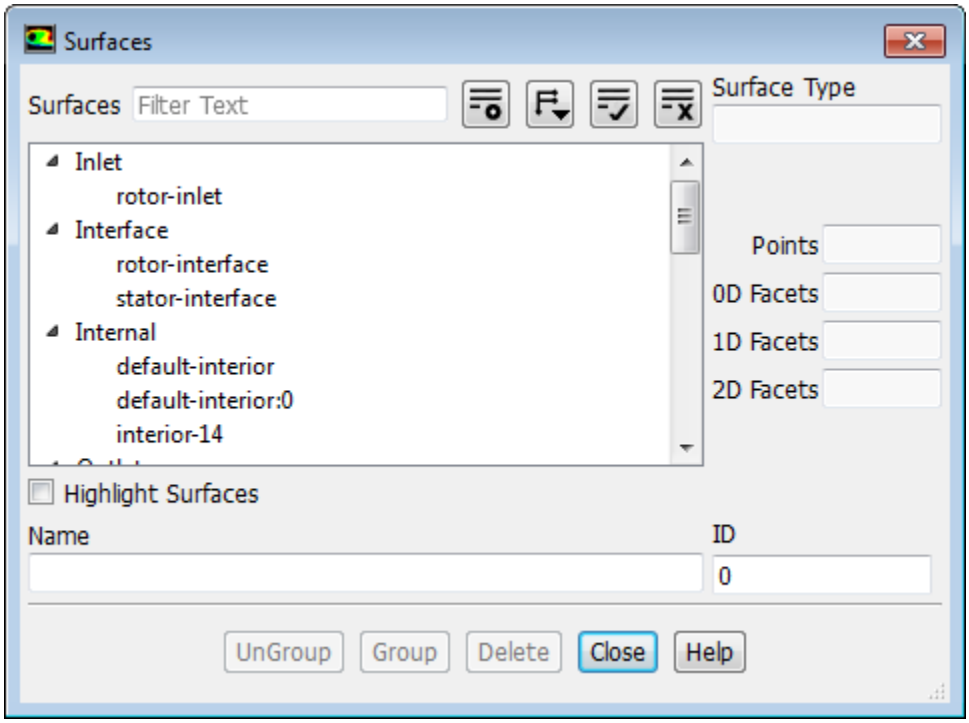
Iso-Clip
opens the Iso-Clip Dialog Box (p. 2720).

OK
creates the report definition.

Compute
computes the value of the report definition at the selected surface(s) and prints to the console.

38.2.87. Surfaces Dialog Box

The **Surfaces** dialog box allows you to interactively group, rename, and delete surfaces and obtain information about their components. See Grouping, Renaming, and Deleting Surfaces (p. 1734) for details about the items below.



Controls

Surfaces
contains a list of existing surfaces from which you can select the surface(s) of interest.

Name
displays the name of the selected surface. You can edit the text field to modify the surface name. (If more than one surface is selected, the name of the first one you selected will be displayed.)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface Type
displays the type of surface that is selected (for example, `zone-surf` if one surface is selected, or `Multiple Surfaces` if more than one surface is selected).

Points, 0D Facets, 1D Facets,
and **2D Facets** display the number of points and facets in the selected surface. If more than one surface is selected, the sum over all selected surfaces is displayed for each quantity.

Note that if you want to check these statistics for a surface that was read from a case file, you will need to first display it.

Highlight Surfaces
when enabled highlights the surfaces (selected in the **Surfaces** dialog box) in the graphics window.

ID
displays the ID of the selected surface. You cannot change this value.

UnGroup
ungroups the selected surface. This button is available only if the selected surface was created by **Group**ing two or more surfaces together.

Rename
renames the selected surface in **Surfaces** with the name specified in **Name**. This button is available when just one surface is selected. (If two or more surfaces are selected, it becomes the **Group** button.)

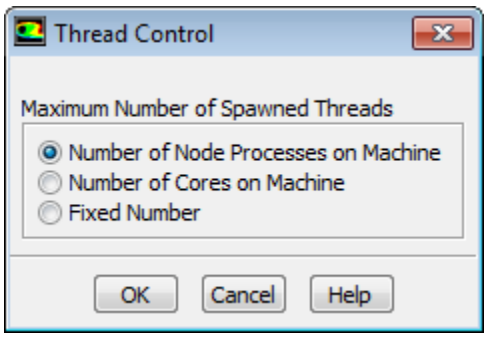
Group
groups two or more selected surfaces and gives the group the name entered in **Name**. This button replaces the **Rename** button when two or more surfaces are selected.

Delete
deletes the selected surface(s).

38.2.88. Thread Control Dialog Box

The **Thread Control** dialog box allows you to control the maximum number of threads on each machine, as described in Controlling the Threads (p. 2058).

Parallel → General → Thread Control...



Controls

Maximum Number of Spawned Threads
contains a list of options for defining the maximum number of threads on each machine.

Number of Node Processes on Machine
specifies that the maximum number of threads on each machine is equal to the number of ANSYS Fluent node processes on each machine.

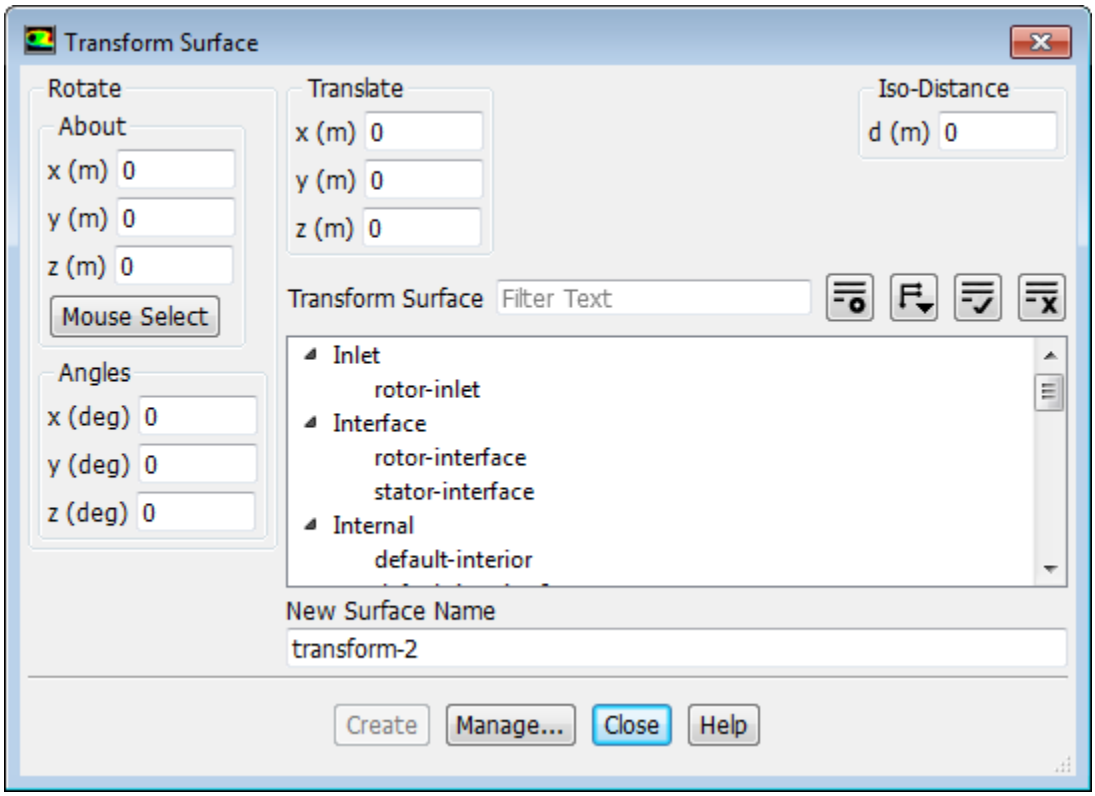
Number of Cores on Machine
specifies that the maximum number of threads on each machine is equal to the number of cores on the machine. ANSYS Fluent obtains the number of cores from the OS.

Fixed Number
specifies that the maximum number of threads that can be spawned on each machine is equal to the number you provide in the number-entry box below **Fixed Number**.

38.2.89. Transform Surface Dialog Box

The **Transform Surface** dialog box allows you to create a new data surface by rotating and/or translating an existing surface, and/or by specifying a constant normal distance from it. See Transforming Surfaces (p. 1732) for details about the items below.

Setting Up Domain → Surface → Create → Transform...



Controls

Rotate
contains the transformation parameters for rotation.

About
defines the origin about which the surface is rotated. You will specify a point, and the origin of the coordinate system for the rotation will be set to the specified point. For example, if you specified the point (1,0) in 2D, rotation would be about the z axis anchored at (1,0). You can either enter the point's

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

coordinates in the **x,y,z** fields or click the **Mouse Select** button and select a point in the graphics window using the mouse.

Angles
define the angles about the **x**, **y**, and **z** axes (that is, the axes of the coordinate system with the origin defined under **About**) by which the surface is rotated. For 2D problems, you can specify rotation about the **z** axis only.

Translate
contains the transformation parameters for translation.

x,y,z
define the distance by which the surface is translated in each direction.

Iso-Distance
contains the transformation parameters for "isodistancing."

d
sets the normal distance between the original surface and the transformed surface.

Transform Surface
contains a list of existing surfaces from which you can select the surface to be transformed. The selected surface will remain unchanged; the transformation will create a new surface.

New Surface Name
designates the name of the new surface. The default is the concatenation of the transformation type (that is, iso-distance, rotate, or translate) and an integer which is the new surface ID.

Create
creates the surface.

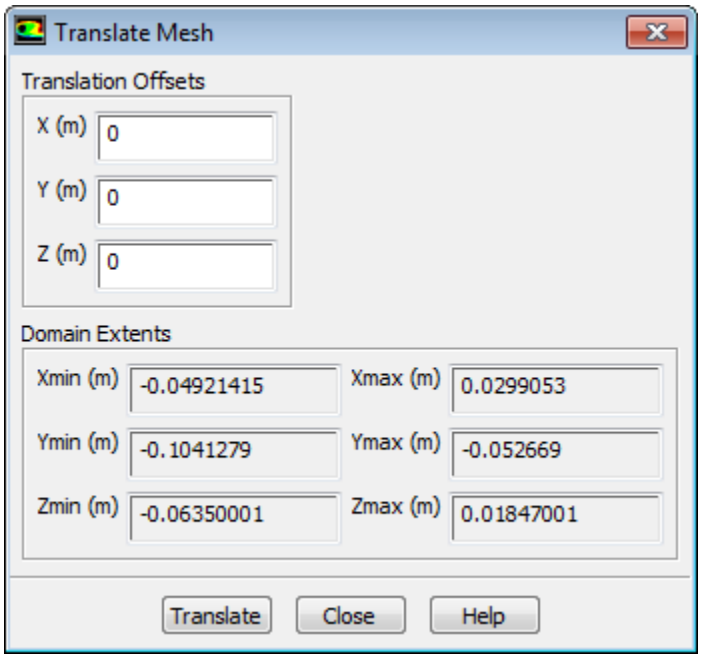
Manage...
opens the Surfaces Dialog Box (p. 2810) in which you can rename and delete surfaces and determine their sizes.

38.2.90. Translate Mesh Dialog Box

The **Translate Mesh** dialog box allows you to change the origin of the mesh. See Translating the Mesh (p. 228) for details.

Setting Up Domain → Mesh → Transform → Translate...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Translation Offsets

contains the desired changes in the mesh coordinates (that is, the desired delta in the axes origin). You can enter a positive or negative real number.

X,Y,Z

defines the deltas in the x, y, and z directions, in the current units of length.

Domain Extents

displays the Cartesian coordinate extremes of the nodes in the mesh. (These values are not editable; they are purely informational.)

Xmin, Ymin, Zmin

shows the minimum values of Cartesian coordinates in the mesh.

Xmax, Ymax, Zmax

shows the maximum values of Cartesian coordinates in the mesh.

Translate

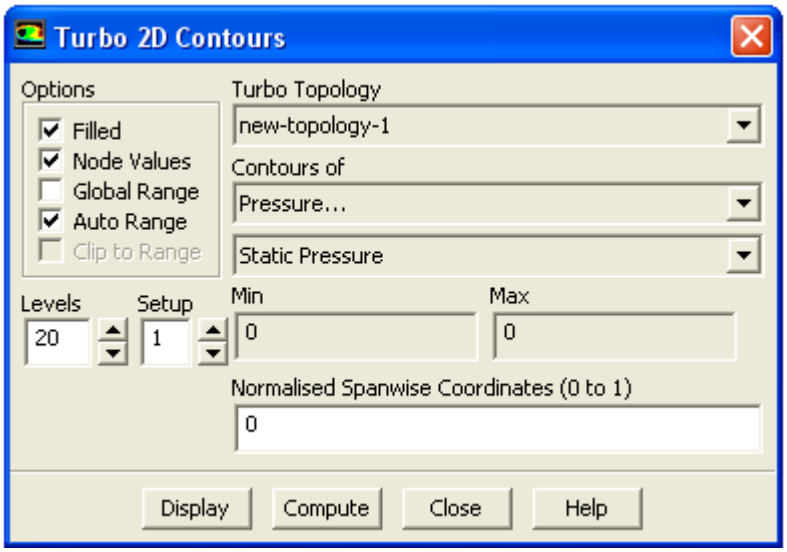
adds the specified translation offsets to the appropriate Cartesian coordinate of every node in the mesh.

38.2.91. Turbo 2D Contours Dialog Box

The **Turbo 2D Contours** dialog box enables you to display turbomachinery-specific contours of variables on surfaces of constant pitchwise, spanwise, or meridional coordinates, projected onto a plane. See Displaying Turbomachinery 2D Contours (p. 1869) for details.

Postprocessing → Model Specific → Turbo Topology → 2D Contours...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Turbo Topology
contains a list of defined topologies. Select from the list to display the values for the selected topology.

Options
contains the check buttons that set various contour display options.

Filled
toggles between filled contours and line contours.

Node Values
toggles between using scalar field values at nodes and at cell centers for computing the contours.

Global Range
toggles between basing the minimum and maximum values on the range of values on the selected surfaces (off), and basing them on the range of values in the entire domain (on, the default).

Auto Range
toggles between automatic and manual setting of the contour range. Any time you change the **Contours of** selection, **Auto Range** is reset to on.

Clip to Range
determines whether or not values outside the prescribed **Min**/ **Max** range are contoured when using **Filled** contours. If selected, values outside the range will not be contoured. If not selected, values below the **Min** value will be colored with the lowest color on the color scale, and values above the **Max** value will be colored with the highest color on the color scale. See Specifying the Range of Magnitudes Displayed (p. 1759) for details.

Levels
sets the number of contour levels that are displayed.

Setup
indicates the ID number of the contour setup. For frequently used combinations of contour fields and options, you can store the information needed to generate the contour plot by specifying a **Setup** number and setting up the desired information in the dialog box. See Storing Contour Plot Settings (p. 1762) for details.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Contours of
contains a list from which you can select the scalar field to be contoured.

Min
shows the minimum value of the scalar field. If **Auto Range** is off, you can set the minimum by typing a new value.

Max
shows the maximum value of the scalar field. If **Auto Range** is off, you can set the maximum by typing a new value.

Normalised Spanwise Coordinates (0 to 1)
enables you to specify the coordinate for the spanwise surface you want to create.

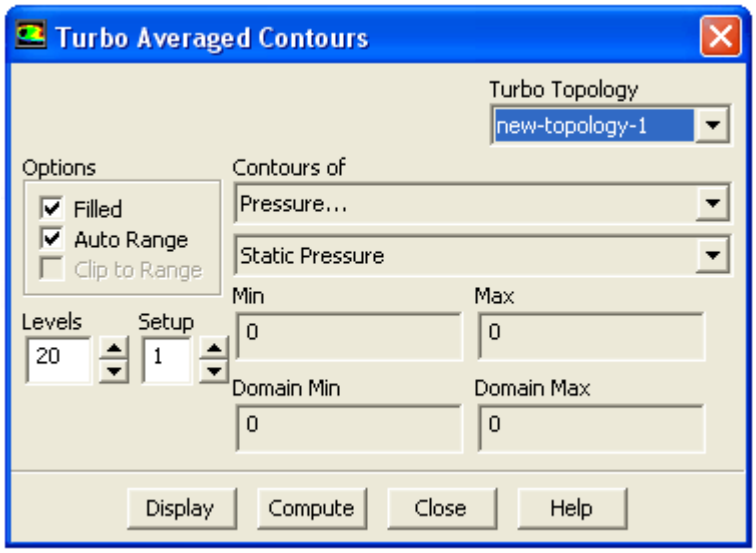
Display
draws the contours in the active graphics window.

Compute
calculates the scalar field and updates the **Min** and **Max** values (even when **Auto Range** is off).

38.2.92. Turbo Averaged Contours Dialog Box

The **Turbo Averaged Contours** dialog box enables you to display turbomachinery-specific circumferentially averaged contours of variables projected on an r- z plane. See Displaying Turbomachinery Averaged Contours (p. 1867) for details.

Postprocessing → Model Specific → Turbo Topology → Averaged Contours...



Controls

Turbo Topology
contains a list of defined topologies. Select from the list to display the values for the selected topology.

Options
contains the check buttons that set various contour display options.

Filled
toggles between filled contours and line contours.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Auto Range
toggles between automatic and manual setting of the contour range. Any time you change the **Contours of** selection, **Auto Range** is reset to on.

Clip to Range
determines whether or not values outside the prescribed **Min**/ **Max** range are contoured when using **Filled** contours. If selected, values outside the range will not be contoured. If not selected, values below the **Min** value will be colored with the lowest color on the color scale, and values above the **Max** value will be colored with the highest color on the color scale. See Specifying the Range of Magnitudes Displayed (p. 1759) for details.

Levels
sets the number of contour levels that are displayed.

Setup
indicates the ID number of the contour setup. For frequently used combinations of contour fields and options, you can store the information needed to generate the contour plot by specifying a **Setup** number and setting up the desired information in the dialog box. See Storing Contour Plot Settings (p. 1762) for details.

Contours of
contains a list from which you can select the scalar field to be contoured.

Min
shows the minimum value of the scalar field. If **Auto Range** is off, you can set the minimum by typing a new value.

Max
shows the maximum value of the scalar field. If **Auto Range** is off, you can set the maximum by typing a new value.

Domain Min
shows the global minimum value of the scalar field for the entire domain.

Domain Max
shows the global maximum value of the scalar field for the entire domain.

Display
draws the contours in the active graphics window.

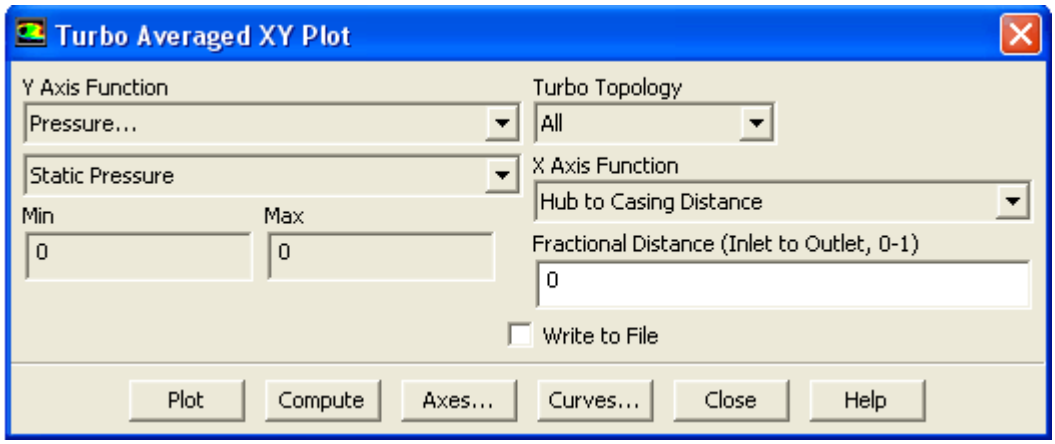
Compute
calculates the scalar field and updates the **Min** and **Max** values (even when **Auto Range** is off).

38.2.93. Turbo Averaged XY Plot Dialog Box

The **Turbo Averaged XY Plot** dialog box enables you to display data in an XY plot format as a function of either the meridional or the spanwise coordinate. See Generating Averaged XY Plots of Turbomachinery Solution Data (p. 1871) for details.

Postprocessing → Model Specific → Turbo Topology → Averaged XY Plot...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Y Axis Function
contains a list of solution variables that can be used for the y axis of the plot.

Min
shows the minimum value of the scalar field.

Max
shows the maximum value of the scalar field.

Turbo Topology
contains a list of defined topologies. Select from the list to display the values for the selected topology.

X Axis Function
enables you to select the coordinate to be used for the x axis of the plot. The choices are **Hub to Casing Distance** (spanwise coordinate), and **Meridional Distance** (meridional coordinate).

Fractional Distance
sets a fractional value (0 to 1) for either the spanwise **Hub to Casing** distance or the meridional **Inlet to Outlet** distance, depending on your selection for **X Axis Function**.

Write to File
enables the file-writing option. When this option is selected, the **Plot** button will change to **Write...**. Clicking on the **Write...** button will open The Select File Dialog Box (p. 19), in which you can specify a name and save a file containing the plot data. The format of this file is described in XY Plot File Format (p. 1850).

Plot
plots the specified surface and/or file data in the active graphics window using the current axis and curve attributes. If the **Write to File** option is turned on, this button becomes the **Write...** button.

Write...
opens The Select File Dialog Box (p. 19), in which you can save the plot data to a file. This button replaces the **Plot** button when the **Write to File** option is turned on.

Compute
calculates the scalar field and updates the **Min** and **Max** values.

Axes...
opens the Axes Dialog Box (p. 2608), which enables you to customize the plot axes.

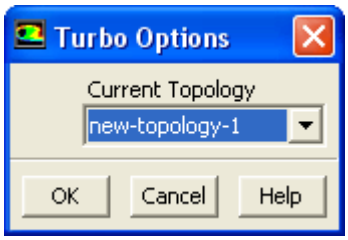
Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Curves...
opens the Curves Dialog Box (p. 2610), which enables you to customize the curves used in the XY plot.

38.2.94. Turbo Options Dialog Box

The **Turbo Options** dialog box enables you to globally set the turbomachinery topology for your model. See Globally Setting the Turbomachinery Topology (p. 1872) for details.

Postprocessing → Model Specific → Turbo Topology → Options...



Controls

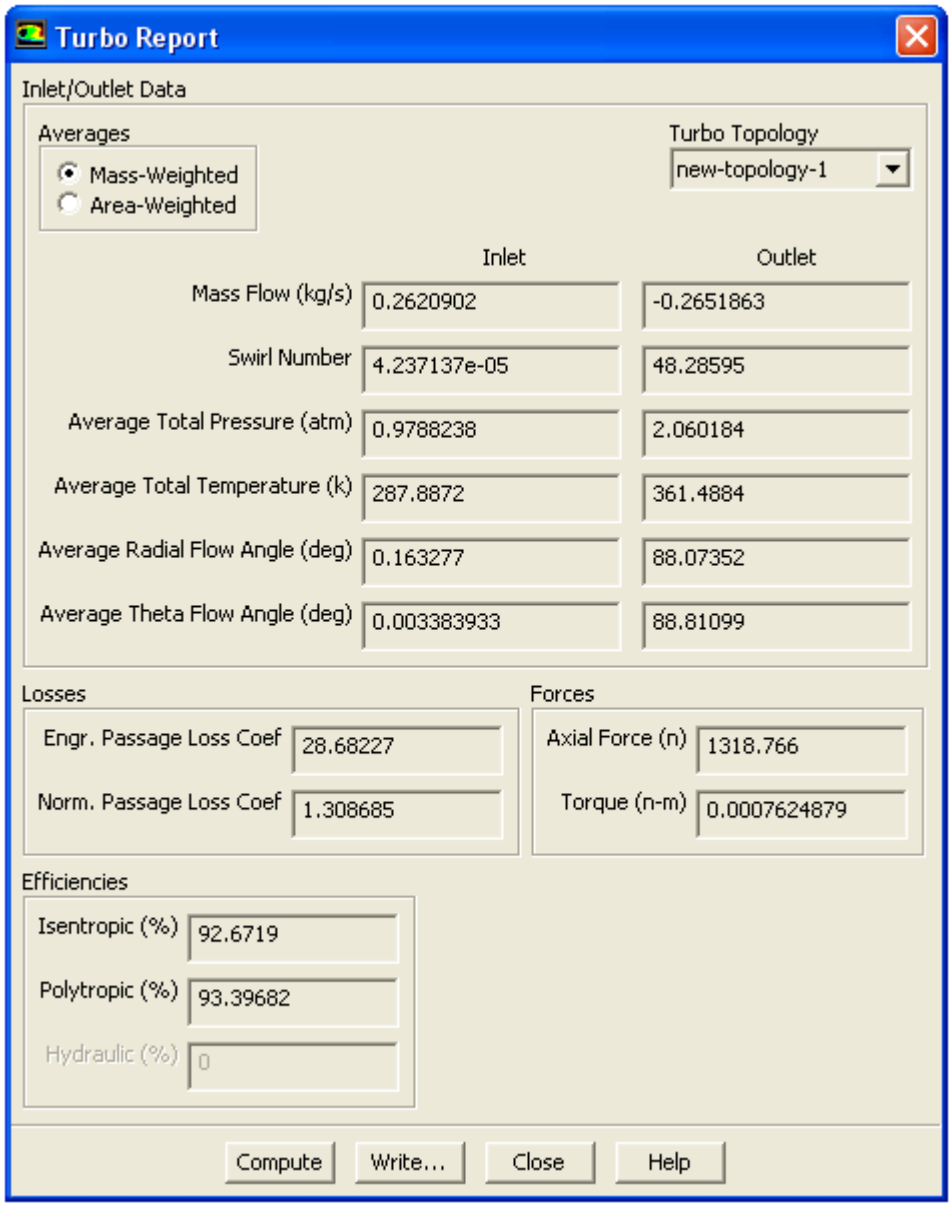
Current Topology
contains a list of predefined turbo topologies.

38.2.95. Turbo Report Dialog Box

The **Turbo Report** dialog box allows you to calculate turbomachinery-specific quantities and integrals. See Generating Reports of Turbomachinery Data (p. 1859) for details.

Postprocessing → Model Specific → Turbo Topology → Report...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Inlet/Outlet Data

contains quantities that can be calculated at inlets and outlets.

Averages

allows you to choose between **Mass-Weighted** and **Area-Weighted** averages for all applicable computed quantities. These quantities are calculated for the **Inlet** and **Outlet** topologies where applicable.

Turbo Topology

contains a list of defined topologies. Select from the list to display the values for the selected topology.

Mass Flow

is the mass flow rate through a surface as defined in Equation 31.2 (p. 1861).

Swirl Number

is the swirl number as defined in Equation 31.3 (p. 1861).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Average Total Pressure
is the area-averaged or mass-averaged total pressure as defined in Equation 31.4 (p. 1861) or Equation 31.5 (p. 1861).

Average Total Temperature
is the area-averaged or mass-averaged total temperature as defined in Equation 31.6 (p. 1862) or Equation 31.7 (p. 1862).

Average Radial Flow Angle
is the area-averaged or mass-averaged radial flow angle as defined in Equation 31.8 (p. 1862) or Equation 31.10 (p. 1863).

Average Theta Flow Angle
is the area-averaged or mass-averaged tangential flow angle as defined in Equation 31.9 (p. 1862) or Equation 31.11 (p. 1863).

Losses
contains the values of loss-related coefficients.

Engr. Passage Loss Coef
is the engineering loss coefficient as defined in Equation 31.12 (p. 1863).

Norm. Passage Loss Coef
is the normalized loss coefficient as defined in Equation 31.13 (p. 1863).

Forces
contains the axial force and the torque on the rotating parts.

Axial Force
is the axial force on the rotating parts as defined in Equation 31.14 (p. 1863).

Torque
is the torque on the rotating parts as defined in Equation 31.15 (p. 1864).

Efficiencies
contains the values of isentropic, polytropic and hydraulic efficiencies.

Isentropic
is the isentropic efficiency for a compressor or a turbine (motor) calculated in the presence of a compressible working fluid as defined in Equation 31.20 (p. 1865) or Equation 31.26 (p. 1867).

Polytropic
is the polytropic efficiency for a compressor or a turbine (motor) calculated in the presence of a compressible working fluid as defined in Equation 31.21 (p. 1866) or Equation 31.27 (p. 1867).

Hydraulic
is the hydraulic efficiency for a pump or a hydraulic turbine (motor) calculated in the presence of an incompressible working fluid as defined in Equation 31.16 (p. 1864) or Equation 31.22 (p. 1866).

Compute
starts the calculation of the quantities in all the fields in the **Turbo Report** dialog box. Note that this process may take some time for a large problem.

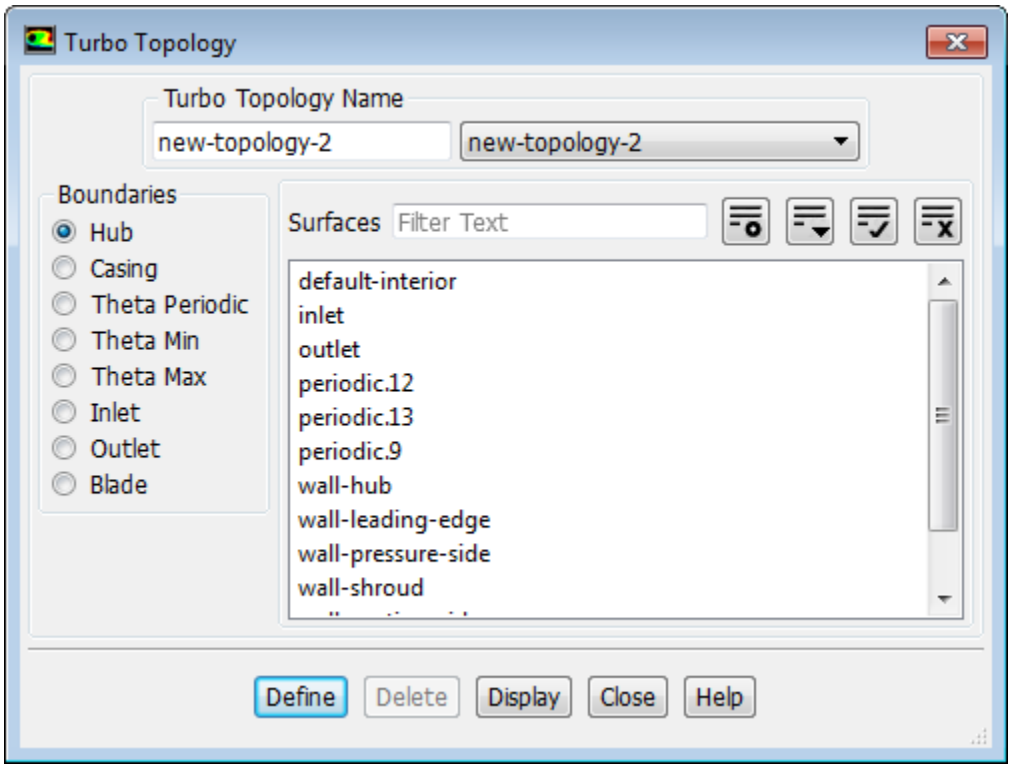
Write...
opens the **Select File** dialog box, which you can use to save the reported values to a file.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.2.96. Turbo Topology Dialog Box

The **Turbo Topology** dialog box allows you to define the topology for a turbomachinery application, so that you can use the turbomachinery-specific postprocessing features described in Turbomachinery Postprocessing (p. 1856). See Defining the Turbomachinery Topology (p. 1856) for details about the items below.

Setting Up Domain → Mesh Models → Turbo Topology...



Controls

Turbo Topology Name
specifies the name of the new topology.

Boundaries
contains radio buttons for the topology boundaries to be defined.

Hub
specifies the definition for the wall zone(s) forming the lower boundary of the flow passage (generally toward the axis of rotation of the machine).

Casing
specifies the definition of the wall zone(s) forming the upper boundary of the flow passage (away from the axis of rotation of the machine).

Theta Periodic
specifies the definition of the periodic boundary zone(s) on the circumferential boundaries of the flow passage.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Theta Min, Theta Max
specify the definition of the wall zones at the minimum and maximum angular (θ) positions on a circumferential boundary.

Inlet
specifies the definition of the inlet zone(s) through which the flow enters the passage.

Outlet
specifies the definition of the outlet zone(s) through which the flow exits the passage.

Blade
specifies the definition of the wall zone(s) that defines the blade(s) (if any). Note that these zones cannot be attached to the circumferential boundaries. For this situation, use **Theta Min** and **Theta Max** to define the blade.

Surfaces
contains a selectable list of the available surfaces, from which you can select the surface(s) that represent the boundary selected under **Boundaries**.

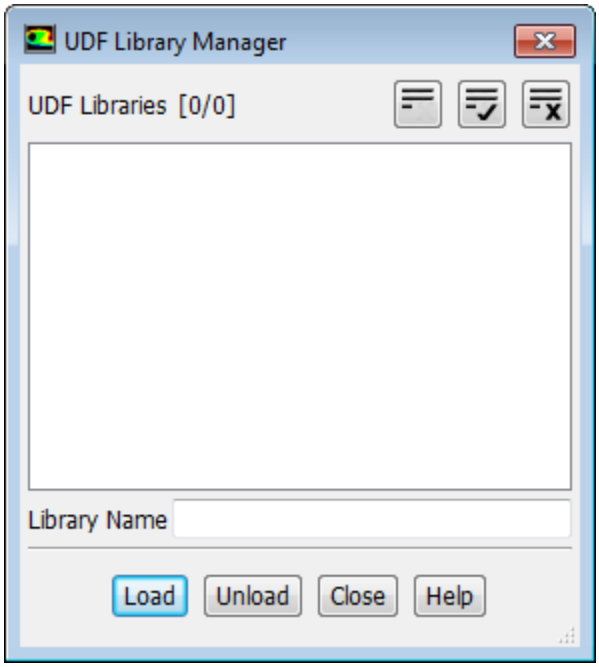
Define
defines the new topology. If you have selected an existing topology the **Define** button is replaced by the **Modify** button.

Display
draws the defined topology in the active graphics window.

38.2.97. UDF Library Manager Dialog Box

The **UDF Library Manager** dialog box allows you to load/unload the user-defined function libraries. See the separate Fluent Customization Manual for details.

User Defined → User Defined → Functions → Manage...



Controls

UDF Libraries
lists the user-defined function libraries that are loaded in ANSYS Fluent.

Library Name
specifies the name of the library to be loaded/unloaded.

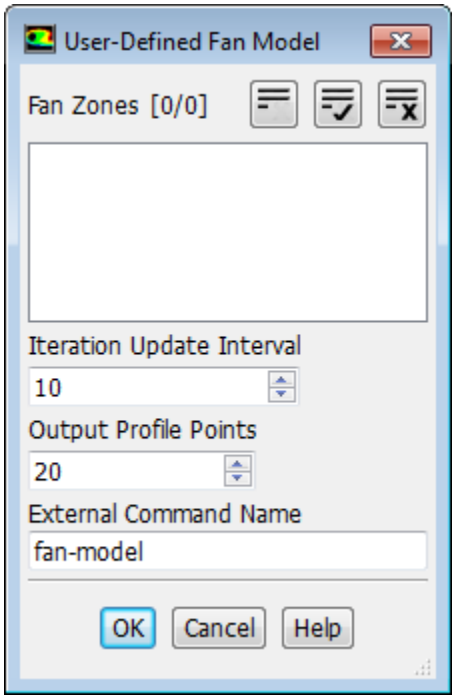
Load
opens the specified library and loads the user-defined function.

Unload
unloads the specified library.

38.2.98. User-Defined Fan Model Dialog Box

The **User-Defined Fan Model** dialog box allows you to periodically regenerate a profile file that can be used to specify the characteristics of a fan, including pressure jump across the fan, and radial and swirling components of velocity generated by the fan. See User-Defined Fan Model (p. 427) for details about this feature and how to use this dialog box.

User Defined → **Model Specific** → **Fan Model...**



Controls

Fan Zones
contains a list from which you can select the fan zone(s) on which your executable will operate.

Iteration Update Interval
specifies how often the executable will be called on to update the fan profile file.

Output Profile Points
specifies the number of points in the profile file to be written by ANSYS Fluent.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

External Command Name
specifies the name of the executable.

38.2.99. User-Defined Function Hooks Dialog Box

The **User-Defined Function Hooks** dialog box allows you to specify user-defined functions (UDFs) connected to a number of models and procedures in ANSYS Fluent. See the separate Fluent Customization Manual for details.

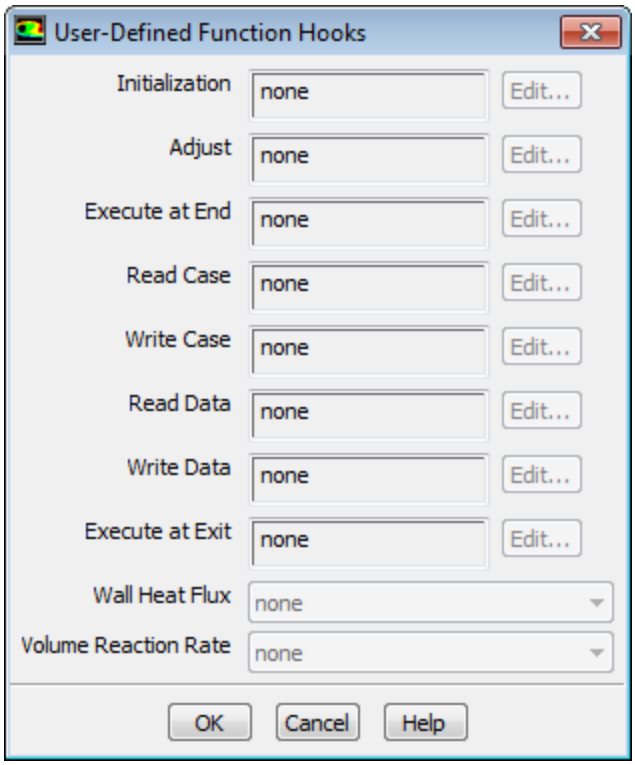
Important

You can hook multiple UDFs to the following functions:

- **Controls**
- **Initialization**
- **Adjust**
- **Execute At End**
- **Read Case**
- **Write Case**
- **Read Data**
- **Write Data**
- **Execute at Exit**

User Defined → **User Defined** → **Function Hooks...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Initialization
selects a UDF that is called immediately after you initialize your flow field.

Adjust
selects a UDF that is called at the beginning of an iteration before solution of velocities, pressure, and other quantities begins.

Execute At End
selects a UDF that is called at the end of an iteration or time-step.

Read Case
selects a UDF that defines a customized section that is to be read from the case file.

Write Case
selects a UDF that defines a customized section that is to be written to the case file.

Read Data
selects a UDF that defines a customized section that is to be read from the data file.

Write Data
selects a UDF that defines a customized section that is to be written to the data file.

Execute at Exit
selects a UDF that is called when exiting an ANSYS Fluent session.

Wall Heat Flux
selects a UDF that modifies the way that the solver computes the heat flux between a wall and the neighboring fluid cells.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Net Reaction Rate
selects a UDF that defines the net reaction rate.

Volume Reaction Rate
selects a UDF that defines a volumetric reaction rate.

Surface Reaction Rate
selects a UDF that defines a surface reaction rate.

Particle Reaction Rate
selects a UDF that defines a particle reaction rate.

Turbulent Premixed Source
selects a UDF that defines the turbulent flame speed and source term for the premixed or partially premixed combustion model.

Chemistry Step
selects a UDF that defines the chemistry step function.

Spray Collide
selects a UDF that defines the spray collide function.

Cavitation Mass Rate
selects a UDF that defines the cavitation rate.

DO Source
selects a UDF that defines the discrete ordinate source function.

DO Diffuse Reflectivity
selects a UDF that defines the diffuse reflectivity function for the DO radiation model.

DO Specular Reflectivity
selects a UDF that defines the specular reflectivity function for the DO radiation model.

Emissivity Weighting Factor
selects a UDF that defines the emissivity weighting factor for the non-gray DO radiation model or the non-gray P-1 radiation model.

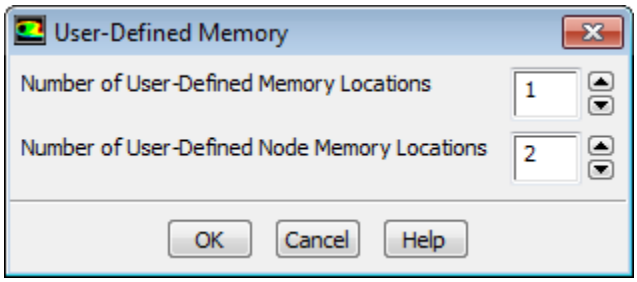
Thickened Flame Model Parameters
selects a UDF that defines the parameters for the Thickened Flame Model.

38.2.100. User-Defined Memory Dialog Box

The **User-Defined Memory** dialog box allows you to allocate memory for user-defined storage variables. See the separate Fluent Customization Manual for details.

User Defined → **User Defined** → **Memory...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Number of User-Defined Memory Locations
specifies the number of memory locations to be allocated.

Number of User-Defined Node Memory Locations
specifies the number of node memory locations to be allocated.

Important

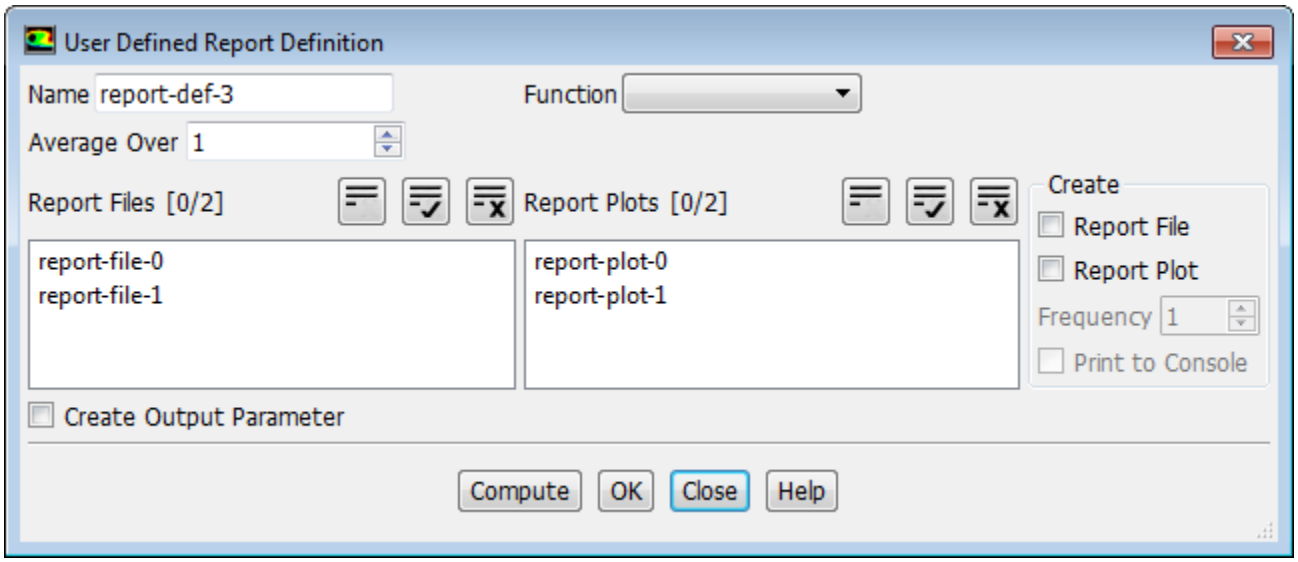
For postprocessing **User-Defined Memory** in CFD-Post, the ANSYS Fluent user must select the UDM quantities using the **Data File Quantities** option and subsequently write the data file to postprocess the quantities in ANSYS Fluent. Alternatively, you have the option of exporting the desired quantities to a `.cdat` file. This will ensure that all the UDM quantities are available for postprocessing in CFD-Post. Refer to the ANSYS CFD-Post manual for more information.

38.2.101. User Defined Report Definition Dialog Box

Solving → Reports → Definitions → New → User Defined...

or

Solution → Report Definitions → New → User Defined



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Function
drop-down list where you can select the function that you loaded for this report definition.

Average Over
(optional) To have Fluent calculate a running average for the **User Defined Report Definition** you can enter a positive integer greater than 1 (the default) for **Average Over**.

Specifying a number greater than 1 means that Fluent will print, plot, and write the running average value of the selected variable instead of the current value of the same variable.

The value reported is averaged over the last N *iterations/time steps, where* N *is your specified* ***Average Over*** *value. When the iteration/time step number is lower than* N*, Fluent calculates the average of the available variable values.*

Report Files
lists all of the report files where you can write the report definition data.

Report Plots
lists all of the report plots where you can plot the report definition data (assuming this report has the same units as the other report definitions included in the selected report plot).

Create

Report File
creates a new report file that includes this report definition.

Report Plot
creates a new report plot that includes this report definition.

Frequency
specifies how frequently the report definition is written, plotted, or printed to the console.

Print to Console
prints the value of the report definition to the console.

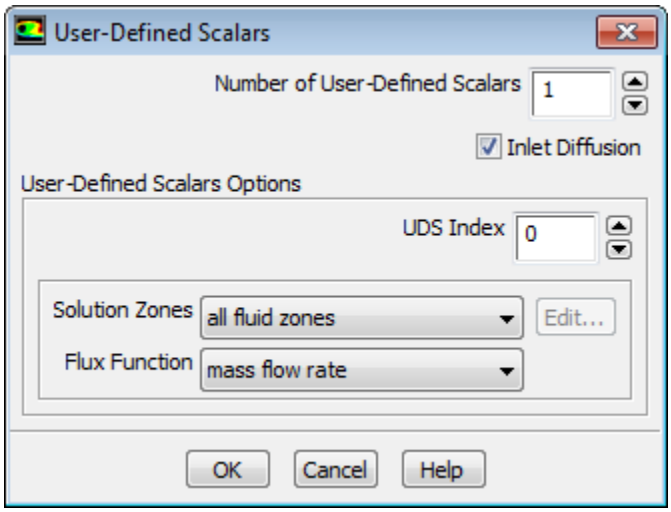
Create Output Parameter
creates an output parameter for this report definition with the name `<report definition name>-op`.

38.2.102. User-Defined Scalars Dialog Box

The **User-Defined Scalars** dialog box allows you to include user-defined scalar transport equations in your calculation. See User-Defined Scalar (UDS) Transport Equations (p. 573) for details.

User Defined → **User Defined** → **Scalars...**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Number of User-Defined Scalars
specifies how many additional scalar transport equations you would like to include in the calculation.

Inlet Diffusion
when enabled allows you to include the diffusion term in the UDS transport equation for all inflow and outflow boundaries.

User-Defined Scalars Options
contains settings that define the scalar transport equation.

UDS Index
when set to 0 marks the first user-defined scalar equation.

Solution Zones
specifies in which zone the scalar equation will be solved: **all fluid zones**, **all solid zones**, **all zones** (fluid and solid) or **selected zones**.

Flux Function
is a drop-down list containing available functions for the convection term of the user-defined scalar transport equation(s).

none
(the default) indicates that there is no convection term included in the scalar transport equation(s); that is, you want to solve a Poisson equation instead of a convection/diffusion equation.

mass flow rate
indicates that the convection term in the scalar transport equation(s) is equal to the mass flow rate $\rho\vec{v}\cdot\vec{A}$.

Note that the **none** and **mass flow rate** options will apply to all solved user-defined scalars. A user-defined flux function must be supplied if a different convective flux is desired for each user-defined scalar.

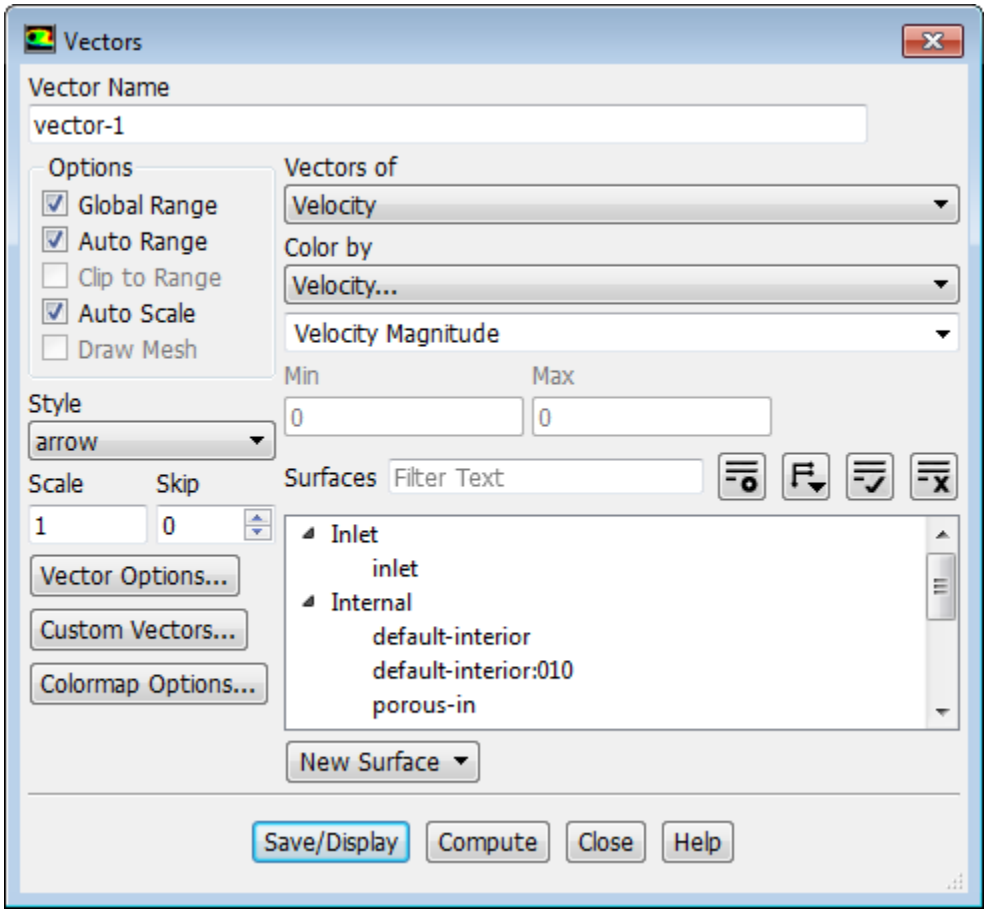
Unsteady Function
is a drop-down list containing available functions for the unsteady term of the user-defined scalar transport equation(s).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.2.103. Vectors Dialog Box

The **Vectors** dialog box controls the display of vector plots. See Displaying Vectors (p. 1764) for details about the items below.

Postprocessing → Graphics → Vectors → New...



Controls

Vector Name
is the name for a vector plot definition. You can specify a name or use the default name **vector-*id***. This control appears only for vector plot definitions.

Options
contains check buttons that set various display options.

Global Range
toggles between basing the minimum and maximum values on the range of values on the selected surfaces (off), and basing them on the range of values in the entire domain (on, the default).

Auto Range
toggles between automatic and manual setting of the range of scalar field values.

Clip to Range
controls the display of vectors that have a value outside the range specified by **Min** and **Max**. When on, no vectors are displayed outside the range. When off, vectors are displayed outside the range using

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the colors at the top and bottom of the color scale. This option is applicable only when **Auto Range** is off. See Specifying the Range of Magnitudes Displayed (p. 1768) for details.

Auto Scale
enables the scaling of all vectors in the domain such that when the **Scale** is 1, there will be minimal overlap of vectors.

Draw Mesh
toggles between displaying and not displaying the mesh. The Mesh Display Dialog Box (p. 2163) is opened when **Draw Mesh** is selected.

Style
selects the style in which the vectors are drawn. Available styles are **cone**, **filled-arrow**, **arrow**, **harpoon**, and **headless**.

Scale
sets the factor by which the vectors should be scaled. See Scaling the Vectors (p. 1766) for details.

Skip
allows you to "thin" or "sample" the vectors that are displayed. See Skipping Vectors (p. 1767) for details.

Vector Options...
opens the Vector Options Dialog Box (p. 2557), in which you can set additional options for vector displays. (This button is not available for vector plot definitions.)

Custom Vectors...
opens the Custom Vectors Dialog Box (p. 2558), in which you can define your own vector fields.

Vectors of
contains a list from which you can select the vector field to be plotted.

Color by
contains a list from which you can select the scalar field by which the vectors are colored.

Min
shows the value to which the lower end of the color scale is mapped. You can set this value manually if **Auto Range** is off.

Max
shows the value to which the upper end of the color scale is mapped. You can set this value manually if **Auto Range** is off.

Surfaces
contains a list from which you can select the surfaces on which to display vectors. In 2D, vectors are displayed on the entire domain if no surface is selected.

New Surface
is a drop-down list button that contains a list of surface options:

Point
opens the Point Surface Dialog Box (p. 2780).

Line/Rake
opens the Line/Rake Surface Dialog Box (p. 2727).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plane
opens the Plane Surface Dialog Box (p. 2777).

Quadric
opens the Quadric Surface Dialog Box (p. 2781).

Iso-Surface
opens the Iso-Surface Dialog Box (p. 2721).

Iso-Clip
opens the Iso-Clip Dialog Box (p. 2720).

Display
draws the vectors in the active graphics window.

Compute
calculates the scalar field and updates the **Min** and **Max** values (even when **Auto Range** is off).

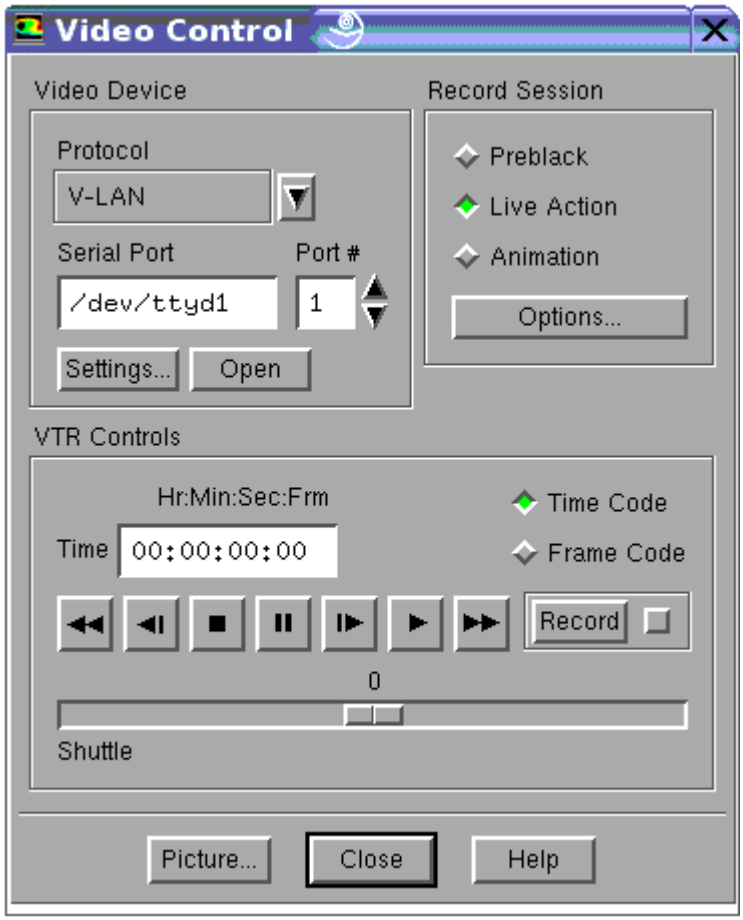
Save/Display
plots the vectors in the active graphics window and saves the vector plot definition. This button appears only for vector plot definitions and replaces the **Display** button.

38.2.104. Video Control Dialog Box

The **Video Control** dialog box is used to record animations or live action to video. When recording animations to video, you must first create your animation (see Creating an Animation (p. 1826)). The **Video Control** dialog box will play your animation and control a videotape recorder (VTR) to record it to video. See Creating Videos (p. 1829) for details about using the video tools.

The **Video Control** dialog box is opened from the Animate Dialog Box (p. 2567).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Video Device

sets up the connection to the VTR control hardware.

Protocol

specifies the command protocol that identifies your VTR controller. ANSYS Fluent currently supports the following VTR controllers:

V-LAN

Machine Control Network is a command protocol developed by Videomedia, Inc. and supported by several videotape editing systems.

MiniVAS

is the command protocol for the MiniVAS and MiniVAS-2 VTR controllers developed by the V.A.S. Group.

To be able to control the video recording from ANSYS Fluent, your VTR controller must use one of these protocols.

Serial Port

identifies the name of the serial port to which your VTR controller is connected.

Port #

is a convenient way of changing the port number associated with the serial port.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Settings...
displays a dialog box that provides options specific to your VTR controller. If your VTR controller protocol is V-LAN, the V-LAN Settings Dialog Box (p. 2836) is displayed; if the protocol is MiniVAS, the MiniVAS Settings Dialog Box (p. 2742) is displayed.

Open
will open a connection to your VTR controller. Once the connection is open, the name of this button will be changed to **Close** so that you can close the connection again.

Record Session
specifies the type of recording you want to perform.

Preblack
is the process of formatting a tape by laying down a time code onto the tape. A tape must be formatted before any frame accurate editing, including frame-by-frame animation, can be performed. During this process, one usually records a black video signal onto the tape as well, thus the name "preblack". When you select this option the current graphics window will be cleared to black. You can use the window to send your black video signal to the VTR. Remember, when you preblack a previously formatted tape, a new time code will be written and any previously recorded video will be destroyed.

Live Action
allows you to record a live ANSYS Fluent session that can be used for demonstration. This option requires your computer's video hardware to have a scan converter that will send the computer display image to your VTR system.

Animation
will play an animation that you have created, and record it onto your VTR system.

Options...
displays the Animation Recording Options Dialog Box (p. 2668), which provides options for recording your animation.

VTR Controls
function like the controls on your VTR's front dialog box.

Time
is a counter that provides input/output of the current tape position. It will update automatically when the tape is repositioned. If you type a new time in this field and press **Enter**, the VTR will go to that position on the tape. For time code, the format is Hrs:Min:Sec:Frames, where the number of frames can range between 0 and 29. For example, a time code of `00:02:36:07` is 2 minutes, 36 seconds, and 7 frames. In order to go to this position on the tape, you can enter the time code as `2:36:07`, leaving out the leading zeros, or you can simply enter `23607`, leaving out the leading zeros and colons. If you select the **Frame Code** radio button, the tape position will be displayed in frames instead of time.

Record
will begin a recording session starting at the current position on the tape. If **Preblack** has been selected as the recording session, the label of this button will be changed to **Preblack**.

Push Button Controls
enable you to operate the VTR's tape transport.

<< rewinds the tape.

<| reverses the tape by one frame.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[ ] places the tape in stop mode.

| | places the tape in pause mode.

|> advances the tape by one frame.

> plays the tape at normal speed.

>> advances the tape in fast-forward mode.

Shuttle
allows you to advance (or reverse) the tape at varying speeds. When you move the shuttle to the right, the tape will be advanced; when you move it to the left, the tape will be reversed. The shuttle speed index ranges from -9 to 9. A speed index of 5 is equivalent to playing the tape at normal speed. A speed index of 0 is equivalent placing the tape in pause mode.

Counter Format
allows you to change the counter units displayed in the **Time/Frame** counter.

Time Code
when selected, displays the tape position as time code.

Frame Code
when selected, displays the tape position as frame code.

This format is for display/user-input purposes only and does not change the actual tape counter format.

Picture...
displays the Picture Options Dialog Box (p. 2776), which provides options for setting the picture size and color levels in the graphics window.

38.2.105. V-LAN Settings Dialog Box

The **V-LAN Settings** dialog box allows you to set specific V-LAN options.

The **V-LAN Settings** dialog box is opened by clicking the **Settings...** button in the Video Control Dialog Box (p. 2833), after you have selected V-LAN as your video device protocol.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

V-LAN Node

identifies the node number of the VTR. This is always 1, unless you have several video devices connected to a V-LAN network, in which case, the VTR you want to use may be identified by another node number.

VTR Video Format

identifies the video format standard to which your VTR conforms. This setting is used to determine the video frame rate (frames/sec).

NTSC

(National Television Standards Committee) is the main video standard used in North America and Japan. The video frame rate is 30 frames/sec.

PAL

(Phase Alternate Line) is the video standard used by some countries in Western Europe, Asia, and Australia. The video frame rate is 25 frames/sec.

Preroll/Postroll

specifies the preroll and postroll times in seconds. These are used when performing **Edit** records, including frame-by-frame animation recording, to make sure the tape is moving at the proper speed through the recorded segment.

Preroll

is the number of seconds the tape will play up to the record in point.

Postroll

is the number of seconds the tape will play past the record out point.

The default settings should be sufficient for most VTR devices.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Live/Real-Time Recording
specifies the type of recording that will be performed during a live action or real-time animation recording session.

Quick Record
can be performed on a tape that has not been formatted or "preblacked". This is also known as "full record". Remember that this option will destroy any pre-existing time code on the tape.

Edit (Preblacked Tape Required)
is used for more precise recording. This mode allows you to record several back-to-back recordings with clean transitions between them.

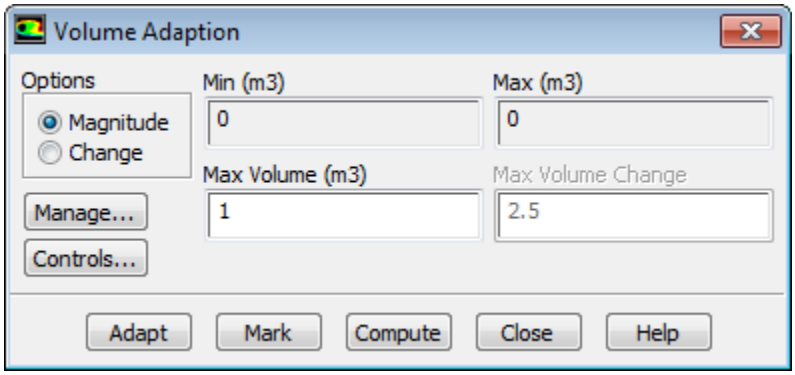
For frame-by-frame animation recording, the record mode used is **Edit** regardless of this setting.

Verbose Mode
enables the echoing of each V-LAN command as it is sent to the V-LAN controller.

38.2.106. Volume Adaption Dialog Box

The **Volume Adaption** dialog box allows you to mark or refine cells based on cell volume or change in cell volume. See Volume Adaption (p. 1694) for details.

Setting Up Domain → Adapt → Mark/Adapt Cells → Volume...



Controls

Options
allows you to specify that the marking and/or refining is based on volume magnitude or volume change.

Magnitude
enables the marking/refining of cells based on volume magnitude.

Change
enables the marking/refining of cells based on the change in volume.

Min
displays the minimum value of cell volume or cell volume change in the mesh. This value is not editable.

Max
displays the maximum value of cell volume or cell volume change in the mesh. This value is not editable.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Max Volume
defines the threshold value for marking/refining the mesh based on volume magnitude. Cells that have volumes greater than the threshold are marked for refinement.

Max Volume Change
defines the threshold value for marking/refining the mesh based on the change in volume. Cells with volume changes that are greater than the threshold value are marked for refinement.

Manage...
opens the Manage Adaption Registers Dialog Box (p. 2728), which allows you to display and manipulate adaption registers that are generated using the **Mark** command.

Controls...
opens the Mesh Adaption Controls Dialog Box (p. 2730), which allows you to control certain aspects of the adaption process.

Adapt
refines the mesh based on either the maximum volume or the volume change, and the adaption limits.

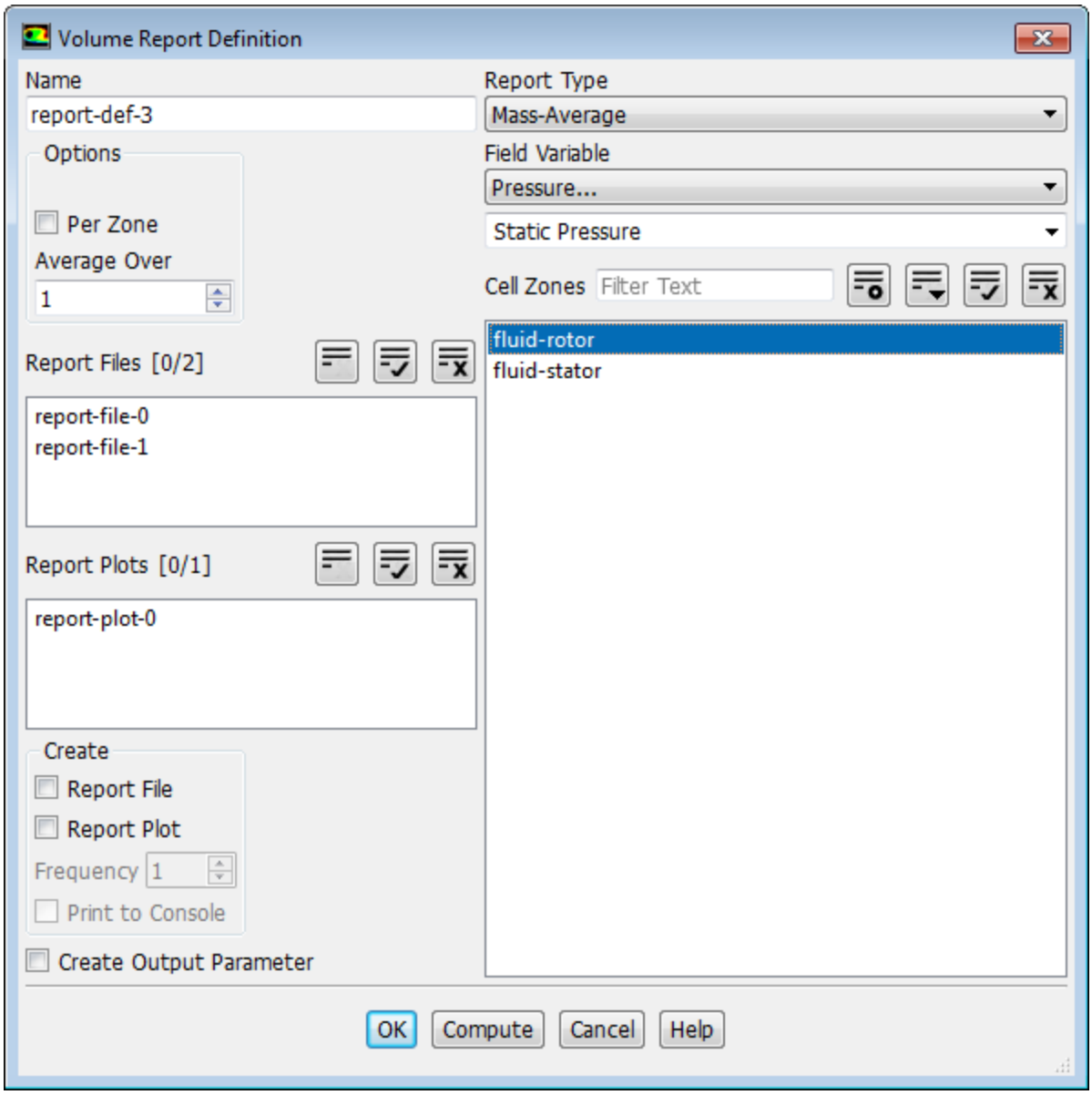
Mark
marks cells to be refined based on either the maximum volume or the volume change. This command produces an adaption register. For information on using adaption registers, see Manipulating Adaption Registers (p. 1701).

Compute
calculates the minimum and maximum cell volume or cell volume change and displays them in the **Min** and **Max** real number fields.

38.2.107. Volume Report Definition Dialog Box

Solving → **Reports** → **Definitions** → **New** → **Volume Report**

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options

Per Zone
specifies whether or not the chosen field variable is calculated from all of the selected cell zones combined (default) or individually on each of the selected cell zones.

Average Over
(optional) To have Fluent calculate a running average for the **Volume Report Definition** you can enter a positive integer greater than 1 (the default) for **Average Over**.

Specifying a number greater than 1 means that Fluent will print, plot, and write the running average value of the selected variable instead of the current value of the same variable.

The value reported is averaged over the last N *iterations/time steps, where* N *is your specified* ***Average Over*** *value. When the iteration/time step number is lower than* N*, Fluent calculates the average of the available variable values.*

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Report Files
lists all of the report files where you can write the report definition data.

Report Plots
lists all of the report plots where you can plot the report definition data (assuming this report has the same units as the other report definitions included in the selected report plot).

Create

Report File
creates a new report file that includes this report definition.

Report Plot
creates a new report plot that includes this report definition.

Frequency
specifies how frequently the report definition is written, plotted, or printed to the console.

Print to Console
prints the value of the report definition to the console.

Create Output Parameter
creates an output parameter for this report definition with the name `<report definition name>-op`.

Report Type
selects the integration method used on the selected cell zones. The available report types are the same as those in the Volume Integrals Dialog Box (p. 2620). See Volume Integration (p. 1922) for details.

Field Variable
contains a list of solution variables that can be monitored on the selected cell zones. This list will not be available if you select **Volume** as the **Report Type**.

Phase
contains a list of all of the phases in the problem that you have defined. This is available when the VOF, mixture, or Eulerian multiphase model is enabled.

Cell Zones
contains a selectable list of the current cell zones.

OK
creates the report definition.

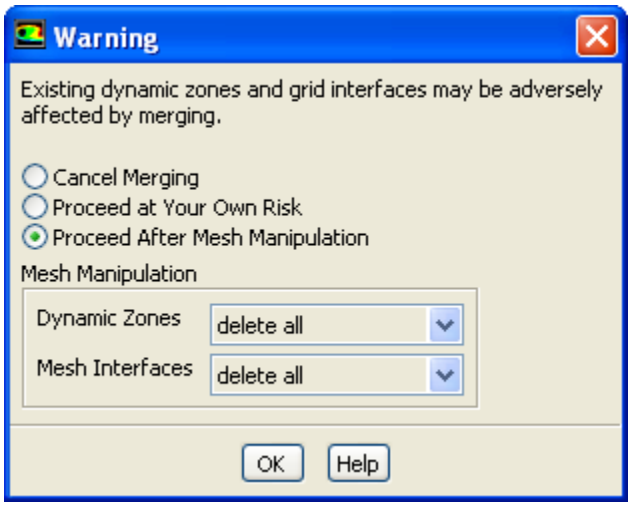
Compute
computes the value of the report definition at the selected zone(s) and prints to the console.

38.2.108. Warning Dialog Box

The **Warning** dialog box allows you to specify whether existing dynamic zones and/or mesh interfaces are deleted when you are using the **Merge Zones** dialog box to merge multiple zones of the same type into a single zone (see Merge Zones Dialog Box (p. 2730) for details). Note that dynamic zones and mesh interfaces that exist at the time of the merge may be adversely affected by the merge.

The **Warning** dialog box opens automatically when you click **Merge** in the Merge Zones Dialog Box (p. 2730) if your case has dynamic zones or mesh interfaces.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Cancel Merging
returns you to the **Mesh Zones** dialog box when you click **OK**, without initiating the merge.

Proceed at Your Own Risk
allows the merge to proceed when you click **OK**, without deleting the existing dynamic zones and mesh interfaces beforehand.

Proceed After Mesh Manipulation
deletes the existing dynamic zones and/or mesh interfaces (based on the settings in the **Mesh Manipulation** group box) and then merges the zones when you click **OK**.

Mesh Manipulation
allows you to specify whether the existing dynamic zones and/or mesh interfaces are deleted prior to the initiation of the merge. This group box is only available when **Proceed After Mesh Manipulation** is selected.

Dynamic Zones
specifies whether all existing dynamic zones are deleted prior to the merge.

Mesh Interfaces
specifies whether all existing mesh interfaces are deleted prior to the merge.

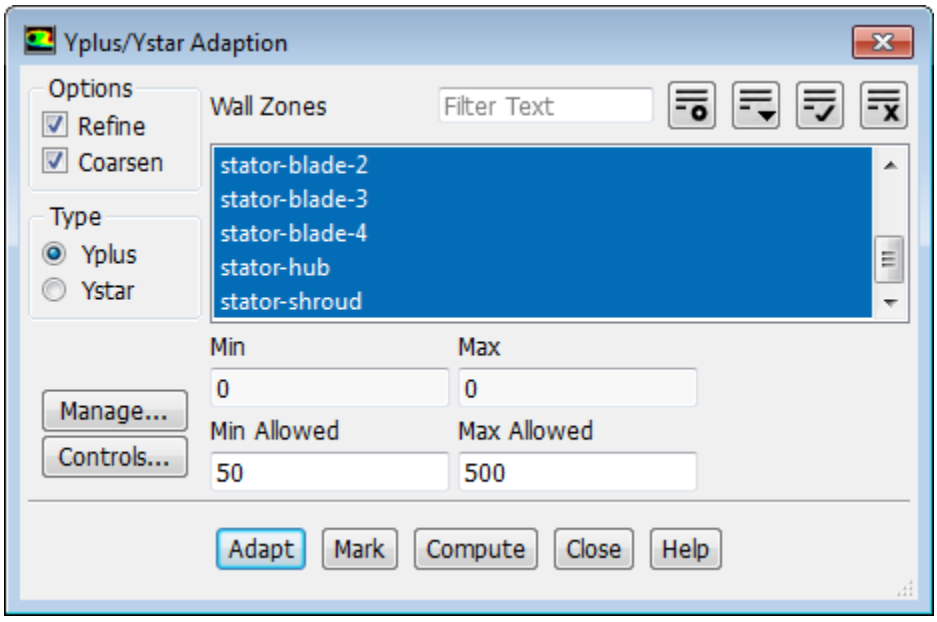
Note that selecting **keep all** from both drop-down lists in the **Mesh Manipulation** group box is the equivalent of selecting **Proceed at Your Own Risk**.

38.2.109. Yplus/Ystar Adaption Dialog Box

The **Yplus/Ystar Adaption** dialog box allows you to mark or adapt boundary cells on specified wall zones based on the non-dimensional y^+ or y^* parameter. See Yplus/Ystar Adaption (p. 1695) for details.

Setting Up Domain → Adapt → Mark/Adapt Cells → Yplus/Ystar...

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options
contains the check buttons that toggle the ability to mark and/or adapt cells for refinement or coarsening.

Refine
toggles the ability to refine cells or mark cells for refinement.

Coarsen
toggles the ability to coarsen cells or mark cells for coarsening.

Type
contains the check buttons that enable adaption based on y^+ or y^*.

Yplus
enables y^+ adaption.

Ystar
enables y^* adaption.

Wall Zones
contains a selectable list of active wall zones. Boundary cells associated with the wall zones you select will be marked or adapted based on the options, thresholds, and limitations applied in the dialog box.

Min/Max
displays the minimum and maximum cell values of y^+ or y^* for all cells associated with viscous wall zones. Note that these values are independent of the wall zones selected. The real number field values are not editable; they are purely informational.

Min Allowed
designates the threshold value for coarsening the mesh. Cells with y^+ or y^* values below the minimum threshold will be marked for coarsening.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Max Allowed
designates the threshold value for refining the mesh. Cells with y^+ or y^* values above the maximum threshold will be marked for refinement.

Manage...
opens the Manage Adaption Registers Dialog Box (p. 2728), which allows you to display and manipulate adaption registers that are generated using the **Mark** command.

Controls...
opens the Mesh Adaption Controls Dialog Box (p. 2730), which allows you to control certain aspects of the adaption process.

Adapt
adapts the mesh based the coarsening and refining toggle buttons, the wall zones selected, the minimum and maximum y^+ or y^* allowed, and the adaption limits.

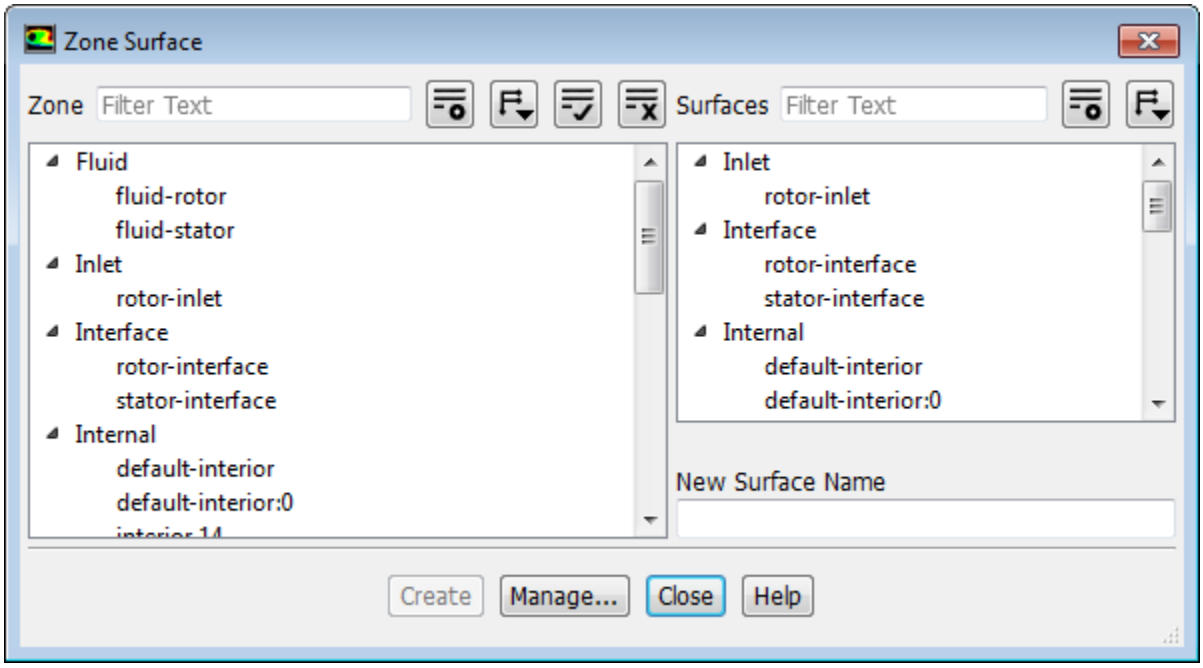
Mark
marks cells to be refined and/or coarsened based on the wall zones selected and the minimum and maximum y^+ or y^* allowed in the mesh. This command produces an adaption register. For information on using adaption registers, see Manipulating Adaption Registers (p. 1701).

Compute
computes the minimum and maximum values of y^+ or y^* on all cells on viscous walls and displays them in the **Min** and **Max** real number fields. Note that these are the extremes for the y^+ or y^* values of every cell on a viscous wall, not just the cells associated with the selected zones.

38.2.110. Zone Surface Dialog Box

The **Zone Surface** dialog box allows you to interactively create surfaces from face and cell zones in the domain. See Zone Surfaces (p. 1710) for details about the items below.

Setting Up Domain → Surface → Create → Zone...



Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Zone
lists zones that you can select for creating zone surfaces.

Surfaces
displays an informational list of existing surfaces.

New Surface Name
designates the name of the surface to be created. The default is the zone name.

Create
creates the surface.

Manage...
opens the Surfaces Dialog Box (p. 2810) in which you can rename and delete surfaces and determine their sizes.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Appendix A. ANSYS Fluent Model Compatibility

The following tables summarize the compatibility of several ANSYS Fluent model categories:

- Multiphase Models (see Modeling Multiphase Flows (p. 1365))
- Moving Domain Models (See Modeling Flows with Moving Reference Frames (p. 603))
- Turbulence Models (See Modeling Turbulence (p. 749))
- Combustion Models (See Chapters Modeling Species Transport and Finite-Rate Chemistry (p. 957) – Modeling Engine Ignition (p. 1139))

Note that a ✓ indicates that two models are compatible with each other, while the absence of a ✓ indicates that two models are not compatible with each other.

Table 1: Moving Domain Models vs. Multiphase Models

	Eulerian	VOF	Mixture	Discrete Phase
Sliding Mesh	✓	✓	✓	✓
Mixing Plane	✓	✓	✓	
Dynamic Mesh	✓	✓	✓	✓
Multiple Reference Frame	✓	✓	✓	✓
Single Reference Frame	✓	✓	✓	✓

Table 2: Multiphase Models vs. Turbulence Models

	Spalart–Allmaras	k-epsilon*	k-omega**	Reynolds Stress	LES
Eulerian		✓	✓	✓	
VOF	✓	✓	✓	✓	✓
Mixture	✓	✓	✓	✓	
Discrete Phase	✓	✓	✓	✓	✓

Table 3: Combustion Models vs. Multiphase Models

	Eulerian	VOF	Mixture	Discrete Phase
Laminar Finite Rate	✓	✓	✓	✓
Eddy Dissipation	✓	✓	✓	✓
Eddy Dissipation Concept				✓

Non-Premixed				✓
Premixed				✓
Partially Premixed				✓
Composition PDF Transport				Eulerian PDF only
Pollutants				✓

Table 4: Moving Domain Models vs. Turbulence Models

	Spalart–Allmaras	k-epsilon*	k-omega**	Reynolds Stress	LES
Sliding Mesh	✓	✓	✓	✓	✓
Mixing Plane	✓	✓	✓	✓	
Dynamic Mesh	✓	✓	✓	✓	✓
Multiple Reference Frame	✓	✓	✓	✓	✓
Single Reference Frame	✓	✓	✓	✓	✓

Table 5: Combustion Models vs. Moving Domain Models

	Sliding Mesh	Mixing Plane	Dynamic Mesh	Multiple Reference Frame	Single Reference Frame
Laminar Finite Rate	✓		✓	✓	✓
Eddy Dissipation	✓		✓	✓	✓
Eddy Dissipation Concept	✓		✓	✓	✓
Non-Premixed	✓		✓	✓	✓
Premixed	✓		✓	✓	✓
Partially Premixed	✓		✓	✓	✓
Composition PDF Transport	✓		✓	✓	✓
Pollutants	✓		✓	✓	✓

Table 6: Combustion Models vs. Turbulence Models

	Spalart-Allmaras	k-epsilon*	k-omega**	Reynolds Stress	LES
Laminar Finite Rate	✓	✓	✓	✓	✓
Eddy Dissipation		✓	✓	✓	✓

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Eddy Dissipation Concept		✓	✓	✓	✓
Non-Premixed		✓	✓	✓	✓
Premixed		✓	✓	✓	✓
Partially Premixed		✓	✓	✓	✓
Composition PDF Transport		✓	✓	✓	✓
Pollutants		✓	✓	✓	✓

Key:

✓ - compatible

* - Includes Standard, RNG, and Realizable k-epsilon models

** - Includes Standard, BSL, and SST k-omega models

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Appendix B. ANSYS Fluent File Formats

This appendix provides information about the following:

B.1. Case and Data File Formats
B.2. Mesh Morpher/Optimizer File Formats
B.3. Shell Conduction Settings File Format
B.4. 3D Fan Curve File Format

B.1. Case and Data File Formats

This section describes the contents and formats of ANSYS Fluent case and data files. After discussing the Guidelines (p. 2851) and Formatting Conventions in Binary and Formatted Files (p. 2851), the section descriptions are grouped according to function:

- Grid Sections (p. 2852) : Creating grids for ANSYS Fluent.
- Other (Non-Grid) Case Sections (p. 2863)
- Data Sections (p. 2865) : Importing solutions into another postprocessor.

The case and data files may contain other sections that are intended for internal use only.

B.1.1. Guidelines

The ANSYS Fluent case and data files are broken into several sections according to the following guidelines:

- Each section is enclosed in parentheses and begins with a decimal integer indicating its type. This integer is different for formatted and binary files (Formatting Conventions in Binary and Formatted Files (p. 2851)).
- All groups of items are enclosed in parentheses. This makes skipping to ends of (sub)sections and parsing them very easy. It also allows for easy and compatible addition of new items in future releases.
- Header information for lists of items is enclosed in separate sets of parentheses preceding the items, and the items are enclosed in their own parentheses.

B.1.2. Formatting Conventions in Binary and Formatted Files

For formatted files, examples of file sections are given in Grid Sections (p. 2852) and Other (Non-Grid) Case Sections (p. 2863). For binary files, the header indices described in this section (for example, `10` for the node section) are preceded by `20` for single-precision binary data, or by `30` for double-precision binary data (for example, `2010` or `3010` instead of `10`). The end of the binary data is indicated by `End of Binary Section 2010` or `End of Binary Section 3010` before the closing parenthesis of the section.

An example with the binary data represented by periods is as follows:

```
(2010 (2 1 2aad 2 3)(
  .
```

```
.
.
)
End of Binary Section 2010)
```

B.1.3. Grid Sections

Grid sections are stored in the case file. A grid file is a subset of a case file, containing only those sections pertaining to the grid. The currently defined grid sections are:

- Comment (See Comment (p. 2852))
- Header (See Header (p. 2853))
- Dimensions (See Dimensions (p. 2853))
- Nodes (See Nodes (p. 2853))
- Periodic Shadow Faces (See Periodic Shadow Faces (p. 2854))
- Cells (See Cells (p. 2855))
- Faces (See Faces (p. 2856))
- Face Tree (See Face Tree (p. 2858))
- Cell Tree (See Cell Tree (p. 2858))
- Interface Face Parents (See Interface Face Parents (p. 2859))

The section ID numbers are indicated in both symbolic and numeric forms. The symbolic representations are available as symbols in a Scheme source file (`xfile.scm`), which is available from ANSYS Inc., or as macros in a C header file (`xfile.h`), which is located in your installation area.

B.1.3.1. Comment

Index:	0
Scheme symbol:	`xf-comment`
C macro:	`XF_COMMENT`
Status:	optional

Comment sections can appear anywhere in the file (except within other sections) as:

```
(0 "comment text")
```

You should precede each long section, or group of related sections, by a comment section explaining what is to follow.

Example:

```
(0 "Variables:")
 (37 (
 (relax-mass-flow 1)
 (default-coefficient ())
 (default-method 0)
 ))
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

B.1.3.2. Header

Index:	1
Scheme symbol:	`xf-header`
C macro:	XF_HEADER
Status:	optional

Header sections can appear anywhere in the file (except within other sections). The following is an example:

```
(0 "fluent18.1.0 build-id: 0")
```

The purpose of this section is to identify the program that wrote the file. Although it can appear anywhere, it is one of the first sections in the file. Additional header sections indicate other programs that may have been used in generating the file. It provides a history mechanism showing where the file came from and how it was processed.

B.1.3.3. Dimensions

Index:	2
Scheme symbol:	`xf-dimension`
C macro:	XF_DIMENSION
Status:	optional

The dimensions of the grid appear as:

```
(2 ND)
```

where ND is 2 or 3. This section is supported as a check that the grid has the appropriate dimension.

B.1.3.4. Nodes

Index:	10
Scheme symbol:	`xf-node`
C macro:	XF_NODE
Status:	required

Format:

```
(10 (zone-id first-index last-index type ND)(
 x1 y1 z1
 x2 y2 z2
 .
 .
 .
 ))
```

- If `zone-id` is zero, this provides the total number of nodes in the grid. `first-index` will then be one, `last-index` will be the total number of nodes *in hexadecimal*, `type` is equal to `1`, `ND` is the dimensionality of the grid, and there are no coordinates following (the parentheses for the coordinates are omitted as well).

  For example: `(10 (0 1 2d5 1 2))`

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If zone-id is greater than zero, it indicates the zone to which the nodes belong. first-index and last-index are the indices of the nodes in the zone, *in hexadecimal*. The values of last-index in each zone must be less than or equal to the value in the declaration section. **Type** is always equal to 1.

  ND is an optional argument that indicates the dimensionality of the node data, where ND is 2 or 3.

  If the number of dimensions in the grid is two, as specified by the node header, then only x and y coordinates are present on each line.

The following is an example of a 2D grid:

```
(10 (1 1 2d5 1 2)(
 1.500000e-01 2.500000e-02
 1.625000e-01 1.250000e-02
   .
   .
   .
 1.750000e-01 0.000000e+00
 2.000000e-01 2.500000e-02
 1.875000e-01 1.250000e-02
 ))
```

Because the grid connectivity is composed of integers representing pointers (see Cells and Faces), using hexadecimal conserves space in the file and provides for faster file input and output. The header indices are in hexadecimal so that they match the indices in the bodies of the grid connectivity sections. The zone-id and type are also in hexadecimal for consistency.

B.1.3.5. Periodic Shadow Faces

Index:	18
Scheme symbol:	xf-periodic-face
C macro:	XF_PERIODIC_FACE
Status:	required only for grids with periodic boundaries

This section indicates the pairings of periodic faces on periodic boundaries. Grids without periodic boundaries do not have sections of this type. The format of the section is as follows:

```
(18 (first-index last-index periodic-zone shadow-zone)(
 f00 f01
 f10 f11
 f20 f21
 .
 .
 .
 ))
```

where

first-index = index of the first periodic face pair in the list

last-index = index of the last periodic face pair in the list

periodic-zone = zone ID of the periodic face zone

shadow-zone = zone ID of the corresponding shadow face zone

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These are in hexadecimal format. The indices in the section body (`f*`) refer to the faces on each of the periodic boundaries (in hexadecimal), the indices being offsets into the list of faces for the grid.

Note

In this case, `first-index` and `last-index` do *not* refer to face indices. They refer to indices in the list of periodic pairs.

Example:

```
(18 (1 2b a c) (
 12 1f
 13 21
 ad 1c2
 .
 .
 .
 ))
```

B.1.3.6. Cells

Index:	12
Scheme symbol:	`xf-cell`
C macro:	XF_CELL
Status:	required

The declaration section for cells is similar to that for nodes.

```
(12 (zone-id first-index last-index type element-type))
```

Again, `zone-id` is zero to indicate that it is a declaration of the total number of cells. If `last-index` is zero, then there are no cells in the grid. This is useful when the file contains only a surface mesh to alert ANSYS Fluent that it cannot be used. In a declaration section, the `type` has a value of zero and the `element-type` is not present.

For example,

```
(12 (0 1 3e3 0))
```

It states that there are 3e3 (hexadecimal) = 995 cells in the grid. This declaration section is required and must precede the regular cell sections.

The `element-type` in a regular cell section header indicates the type of cells in the section, as follows:

`element-type`	*description*	*nodes/cell*	*faces/cell*
0	mixed		
1	triangular	3	3
2	tetrahedral	4	4
3	quadrilateral	4	4
4	hexahedral	8	6
5	pyramid	5	5
6	wedge	6	5
7	polyhedral	NN	NF

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where NN and NF will vary, depending on the specific polyhedral cell.

Regular cell sections have no body, but they have a header of the same format where `first-index` and `last-index` indicate the range for the particular zone, `type` indicates whether the cell zone is an active zone (solid or fluid), or inactive zone (currently only parent cells resulting from hanging node adaption). Active zones are represented with `type`=1, while inactive zones are represented with `type`=32.

In the earlier versions of ANSYS Fluent, a distinction was made between solid and fluid zones. This is now determined by properties (that is, material type).

A `type` of zero indicates a dead zone and will be skipped by ANSYS Fluent. If a zone is of mixed type (`element-type`=0), it will have a body that lists the `element-type` of each cell.

Example:

```
(12 (9 1 3d 0 0)(
 1 1 1 3 3 1 1 3 1
 .
 .
 .
 ))
```

Here, there are 3D (hexadecimal) = 61 cells in cell zone 9, of which the first 3 are triangles, the next 2 are quadrilaterals, and so on.

B.1.3.7. Faces

Index:	13
Scheme symbol:	`xf-face`
C macro:	XF_FACE
Status:	required

The format for face sections is as follows:

```
(13 (zone-id first-index last-index bc-type face-type))
```

where

`zone-id` = zone ID of the face section
`first-index` = index of the first face in the list
`last-index` = index of the last face in the list
`bc-type` = ID of the boundary condition represented by the face section
`face-type` = ID of the type(s) of face(s) in the section

The current valid boundary condition types are defined in the following table:

`bc-type`	*description*
2	interior
3	wall
4	pressure-inlet, inlet-vent, intake-fan
5	pressure-outlet, exhaust-fan, outlet-vent

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7	symmetry
8	periodic-shadow
9	pressure-far-field
10	velocity-inlet
12	periodic
14	fan, porous-jump, radiator
20	mass-flow-inlet, mass-flow-outlet
24	interface
31	parent (hanging node)
36	outflow
37	axis

The faces resulting from the intersection of non-conformal grids are placed in a separate face zone, where a factor of 1000 is added to the `bc-type` (for example, 1003 is a wall zone).

The current valid face types are defined in the following table:

`face-type`	*description*	*nodes/face*
0	mixed	
2	linear	2
3	triangular	3
4	quadrilateral	4
5	polygonal	NN

where NN will vary, depending on the specific polygonal face.

A `zone-id` of zero indicates a declaration section, which provides a count of the total number of faces in the file. Such a section omits the `bc-type` and is not followed by a body with further information.

A nonzero `zone-id` indicates a regular face section, and will be followed by a body that contains information about the grid connectivity. Each line of the body will describe one face and will have the following format:

```
n0 n1 n2 c0 c1
```

where,

`n*` = defining nodes (vertices) of the face

`c*` = adjacent cells

This is the format for a 3D grid with a triangular face format. The actual number of nodes depends on the `face-type`. The order of the cell indices is important, and is determined by the right-hand rule: if you curl the fingers of your right hand in the order of the nodes, your thumb will point toward `c0`.

For 2D grids, `n2` is omitted. `c1` is determined by the cross product of two vectors, $\hat{r}$ and $\hat{k}$. The $\hat{r}$ vector extends from `n0` to `n1`, whereas the $\hat{k}$ vector has its origin at `n0` and points out of the grid plane toward the viewer. If you extend your right hand along $\hat{r}$ and curl your fingers in the direction of the angle between $\hat{r}$ and $\hat{k}$, your thumb will point along $\hat{r}\times\hat{k}$ toward `c1`.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the face zone is of mixed type (face-type= 0) or of polygonal type (face-type= 5), each line of the section body will begin with a reference to the number of nodes that make up that particular face, and has the following format:

```
x n0 n1 ... nf c0 c1
```

where,

x = the number of nodes (vertices) of the face

nf = the final node of the face

All cells, faces, and nodes have positive indices. If a face has a cell only on one side, then either c0 or c1 is zero. For files containing only a surface mesh, both these values are zero.

For information on face-node connectivity for various cell types in ANSYS Fluent, refer to Face-Node Connectivity in ANSYS Fluent (p. 121).

B.1.3.8. Face Tree

Index:	59
Scheme symbol:	xf-face-tree
C macro:	XF_FACE_TREE
Status:	only for grids with hanging-node adaption

This section indicates the face hierarchy of the grid containing hanging nodes. The format of the section is as follows:

```
(59 (face-id0 face-id1 parent-zone-id child-zone-id)
(
number-of-kids kid-id-0 kid-id-1 ... kid-id-n
.
.
.
))
```

where,

face-id0 = index of the first parent face in the section

face-id1 = index of the last parent face in the section

parent-zone-id = ID of the zone containing parent faces

child-zone-id = ID of the zone containing children faces

number-of-kids = the number of children of the parent face

kid-id-n = the face IDs of the children

These are in hexadecimal format.

B.1.3.9. Cell Tree

Index:	58
Scheme symbol:	xf-cell-tree
C macro:	XF_CELL_TREE
Status:	only for grids with hanging-node adaption

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This section indicates the cell hierarchy of the grid containing hanging nodes. The format of the section is as follows:

```
(58 (cell-id0 cell-id1 parent-zone-id child-zone-id)
(
number-of-kids kid-id-0 kid-id-1 ... kid-id-n
.
.
.
))
```

where,

`cell-id0` = index of the first parent cell in the section

`cell-id1` = index of the last parent cell in the section

`parent-zone-id` = ID of the zone containing parent cells

`child-zone-id` = ID of the zone containing children cells

`number-of-kids` = the number of children of the parent cell

`kid-id-n` = the cell IDs of the children

These are in hexadecimal format.

B.1.3.10. Interface Face Parents

Index:	61
Scheme symbol:	`xf-face-parents`
C macro:	XF_FACE_PARENTS
Status:	only for grids with non-conformal interfaces

This section indicates the relationship between the intersection faces and original faces. The intersection faces (children) are produced from intersecting two non-conformal surfaces (parents) and are some fraction of the original face. Each child will refer to at least one parent. The format of the section is as follows:

```
(61 (face-id0 face-id1)
(
parent-id-0 parent-id-1
.
.
.
))
```

where,

`face-id0` = index of the first child face in the section

`face-id1` = index of the last child face in the section

`parent-id-*` = index of parent faces

These are in hexadecimal format.

If you set up and save a non-conformal mesh in the solution mode of Fluent and then read it using the meshing mode of Fluent, this section will be skipped; consequently, all the information necessary to

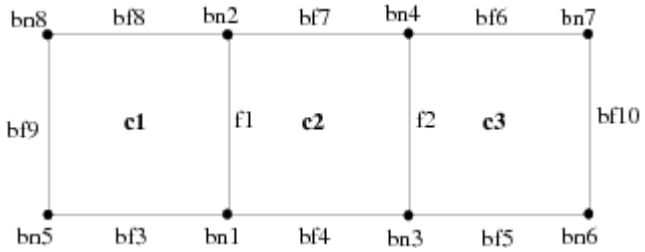
preserve the non-conformal interface will not be maintained. When you switch to or read the mesh back into the solution mode, you will need to recreate the interface.

B.1.3.11. Example Files

B.1.3.11.1. Example 1

Figure 1: Quadrilateral Mesh (p. 2860) illustrates a simple quadrilateral mesh with no periodic boundaries or hanging nodes.

Figure 1: Quadrilateral Mesh



The following describes this mesh:

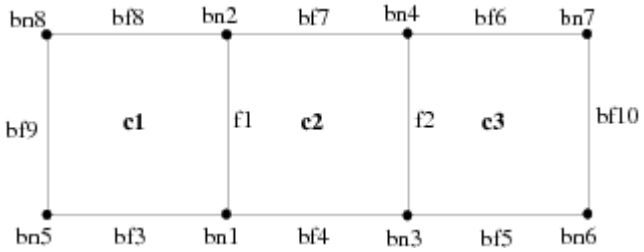
```
(0 "Grid:")

(0 "Dimensions:")
(2 2)

(12 (0 1 3 0))
(13 (0 1 a 0))
(10 (0 1 8 0 2))

(12 (7 1 3 1 3))

(13 (2 1 2 2 2)(
1 2 1 2 3 4 2 3))

(13 (3 3 5 3 2)(
5 1 1 0 1 3 2 0
3 6 3 0))

(13 (4 6 8 3 2)(
7 4 3 0 4 2 2 0
2 8 1 0))

(13 (5 9 9 a 2)(
8 5 1 0))

(13 (6 a a 24 2)(
6 7 3 0))

(10 (1 1 8 1 2)
(
1.00000000e+00 0.00000000e+00
1.00000000e+00 1.00000000e+00
2.00000000e+00 0.00000000e+00
2.00000000e+00 1.00000000e+00
0.00000000e+00 0.00000000e+00
3.00000000e+00 0.00000000e+00
3.00000000e+00 1.00000000e+00
0.00000000e+00 1.00000000e+00))
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

B.1.3.11.2. Example 2

Figure 2: Quadrilateral Mesh with Periodic Boundaries (p. 2861) illustrates a simple quadrilateral mesh with periodic boundaries but no hanging nodes. In this example, bf9 and bf10 are faces on the periodic zones.

Figure 2: Quadrilateral Mesh with Periodic Boundaries



The following describes this mesh:

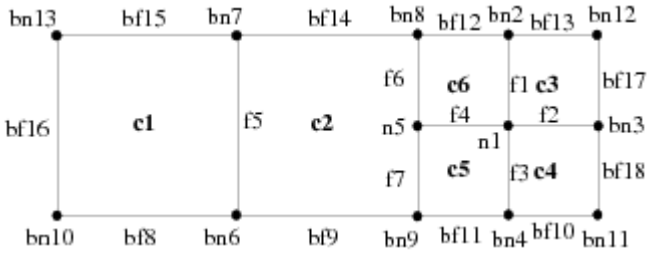
```
(0 "Dimensions:")
(2 2)

(0 "Grid:")

(12 (0 1 3 0))
(13 (0 1 a 0))
(10 (0 1 8 0 2))

(12 (7 1 3 1 3))

(13 (2 1 2 2 2)(
1 2 1 2 3 4 2 3))

(13 (3 3 5 3 2)(
5 1 1 0 1 3 2 0
3 6 3 0))

(13 (4 6 8 3 2)(
7 4 3 0 4 2 2 0
2 8 1 0))

(13 (5 9 9 c 2)(
8 5 1 0))

(13 (1 a a 8 2)(
6 7 3 0))

(18 (1 1 5 1)(
9 a))

(10 (1 1 8 1 2)(
1.00000000e+00 0.00000000e+00
1.00000000e+00 1.00000000e+00
2.00000000e+00 0.00000000e+00
2.00000000e+00 1.00000000e+00
0.00000000e+00 0.00000000e+00
3.00000000e+00 0.00000000e+00
3.00000000e+00 1.00000000e+00
0.00000000e+00 1.00000000e+00))
```

B.1.3.11.3. Example 3

Figure 3: Quadrilateral Mesh with Hanging Nodes (p. 2862) illustrates a simple quadrilateral mesh with hanging nodes.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3: Quadrilateral Mesh with Hanging Nodes



The following describes this mesh:

```
(0 "Grid:")

(0 "Dimensions:")
(2 2)

(12 (0 1 7 0))
(13 (0 1 16 0))
(10 (0 1 d 0 2))

(12 (7 1 6 1 3))
(12 (1 7 7 20 3))

(58 (7 7 1 7)(
4 6 5 4 3))

(13 (2 1 7 2 2)(
1 2 6 3
1 3 3 4
1 4 4 5
1 5 5 6
6 7 1 2
5 8 2 6
9 5 2 5))

(13 (3 8 b 3 2)(
a 6 1 0
6 9 2 0  4 b 4 0
9 4 5 0))

(13 (4 c f 3 2)(
2 8 6 0
c 2 3 0
8 7 2 0
7 d 1 0))

(13 (5 10 10 a 2)(
d a 1 0))

(13 (6 11 12 24 2)(
3 c 3 0
b 3 4 0))

(13 (b 13 13 1f 2)(
c 8 7 0))

(13 (a 14 14 1f 2)(
b c 7 0))

(13 (9 15 15 1f 2)(
9 b 7 0))

(13 (8 16 16 1f 2)(
9 8 2 7))

(59 (13 13 b 4)(
2 d c))
```


Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
(59 (14 14 a 6)(
2 12 11))

(59 (15 15 9 3)(
2 b a))

(59 (16 16 8 2)(
2 7 6))

(10 (1 1 d 1 2)
(
2.50000000e+00 5.00000000e-01
2.50000000e+00 1.00000000e+00
3.00000000e+00 5.00000000e-01
2.50000000e+00 0.00000000e+00
2.00000000e+00 5.00000000e-01
1.00000000e+00 0.00000000e+00
1.00000000e+00 1.00000000e+00
2.00000000e+00 1.00000000e+00
2.00000000e+00 0.00000000e+00
0.00000000e+00 0.00000000e+00
3.00000000e+00 0.00000000e+00
3.00000000e+00 1.00000000e+00
0.00000000e+00 1.00000000e+00))
```

B.1.4. Other (Non-Grid) Case Sections

The following sections store boundary conditions, material properties, and solver control settings.

B.1.4.1. Zone
B.1.4.2. Partitions

B.1.4.1. Zone

Index:	39 or 45
Scheme symbol:	`xf-rp-tv`
C macro:	XF_RP_TV
Status:	required

There is typically one zone section for each zone referenced by the grid. Although some grid zones may not have corresponding zone sections, there cannot be more than one zone section for each zone.

A zone section has the following form:

```
(39 (zone-id zone-type zone-name domain-id)(
(condition1 . value1)
(condition2 . value2)
(condition3 . value3)
.
.
.
 ))
```

Grid generators and other preprocessors need only provide the section header and leave the list of conditions empty, as in

```
(39 (zone-id zone-type zone-name domain-id)())
```

The empty parentheses at the end are required. The solver adds conditions as appropriate, depending on the zone type. When only `zone-id`, `zone-type`, `zone-name`, and `domain-id` are specified, the index 45 is preferred for a zone section. However, the index 39 must be used if boundary conditions

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

are present, because any and all remaining information in a section of index 45 after zone-id, zone-type, zone-name, and domain-id will be ignored.

Here the zone-id is in *decimal* format. This is in contrast to the use of hexadecimal in the grid sections.

The zone-type is one of the following:

```
axis
exhaust fan
fan
fluid
inlet vent
intake fan
interface
interior
mass-flow-inlet
mass-flow-outlet
outlet vent
outflow
periodic
porous-jump
pressure-far-field
pressure-inlet
pressure-outlet
radiator
shadow
solid
symmetry
velocity-inlet
wall
```

The interior, fan, porous-jump, and radiator types can be assigned only to zones of faces inside the domain. The interior type is used for the faces within a cell zone; the others are for interior faces that form infinitely thin surfaces within the domain. ANSYS Fluent allows the wall type to be assigned to face zones both on the inside and on the boundaries of the domain. Some zone types are valid only for certain types of grid components. For example, cell (element) zones can be assigned only one of the following types:

```
fluid
 solid
```

All of the other types listed above can be used only for boundary (face) zones.

The zone-name is a user-specified label for the zone. It must be a valid Scheme symbol [1] and is written without quotes. The rules for a valid zone-name (Scheme symbol) are as follows:

- The first character must be a lowercase letter [2] or a special-initial.
- Each subsequent character must be a lowercase letter, a special-initial, a digit, or a special-subsequent.

where a special-initial character is one of the following:

```
! $ % & * / : < = > ? ~ _ ^
```

and a special-subsequent is one of the following:

```
. + -
```

[1] See Revised (4) Report on the Algorithmic Language Scheme, William Clinger and Jonathan Rees (Editors), 2 November 1991, Section 7.1.1.

[2] The Standard actually only requires that case be insignificant; the ANSYS Fluent implementation accomplishes this by converting all uppercase input to lowercase.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Examples of valid zone names are `inlet-port/cold!—, eggs/easy`, and `e=m*c^2`.

Some examples of zone sections produced by grid generators and preprocessors are as follows:

```
(39 (1 fluid fuel 1)())
 (39 (8 pressure-inlet pressure-inlet-8 2)())
 (39 (2 wall wing-skin 3)())
 (39 (3 symmetry mid-plane 1)())
```

The `domain-id` is an integer that appears after the zone name, associating the boundary condition with a particular phase or mixture (sometimes referred to as phase-domains and mixture-domains).

B.1.4.2. Partitions

Index:	40
Scheme symbol:	`xf-partition`
C macro:	`XF_PARTITION`
Status:	only for partitioned grids

This section indicates each cell's partition. The format of the section is as follows:

```
(40 (zone-id first-index last-index partition-count)(
p1
p2
p3
.
.
.
pn
))
```

where,

`p1` = the partition of the cell whose ID is `first-index`

`p2` = the partition of the cell whose ID is `first-index` +1, and so on

`pn` = the partition of the cell whose ID is `last-index`

`partition-count` = the total number of partitions

Partition IDs must be between 0 and one less than `partition-count`.

B.1.5. Data Sections

The following sections store iterations, residuals, and data field values.

B.1.5.1. Grid Size
B.1.5.2. Data Field
B.1.5.3. Residuals

B.1.5.1. Grid Size

Index:	33
Scheme symbol:	`xf-grid-size`
C macro:	`XF_GRID_SIZE`
Status:	optional

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This section indicates the number of cells, faces, and nodes in the grid that corresponds to the data in the file. This information is used to check that the data and grid match. The format is

```
(33 (n-elements n-faces n-nodes))
```

where the integers are written in decimal.

B.1.5.2. Data Field

Index:	300
Scheme symbol:	`xf-rf-seg-data`
C macro:	XF_RF_SEG_DATA
Status:	required

This section lists a flow field solution variable for a cell or face zone. The data are stored in the same order as the cells or faces in the case file. Separate sections are written out for each variable for each face or cell zone on which the variable is stored. The format is

```
(300 (sub-section-id zone-id size n-time-levels
 n-phases first-id last-id)
 ( data for cell or face with id = first-id
 data-for-cell-or-face with id = first-id+1
 ..
 data-for-cell-or-face with id = last-id
 ))
```

where `sub-section-id` is a (decimal) integer that identifies the variable field (for example, 1 for pressure, 2 for velocity). The complete list of these is available in the header file (`xfile.h`), which is located in your installation area.

where,

`zone-id` = the ID number of the cell or face zone

`size` = the length of the variable vector

`zone-id` matches the ID used in case file. `size` is 1 for a scalar, 2 or 3 for a vector, equal to the number of species for variables defined for each species). `n-time-levels` currently are not used.

A sample data file section for the velocity field in a cell zone for a steady-state, single-phase, 2D problem is shown below:

```
(300 (2 16 2 0 0 17 100)
(8.08462024e-01 8.11823010e-02
 8.78750622e-01 3.15509699e-02
 1.06139672e+00 -3.74040119e-02
...
 1.33301604e+00 -5.04243895e-02
 6.21703446e-01 -2.46118382e-02
 4.41687912e-01 -1.27046436e-01
 1.03528820e-01 -1.01711005e-01
 ))
```

The variables that are listed in the data file depend on the models active at the time the file is written. Variables that are required by the solver based on the current model settings but are missing from the data file are set to their default values when the data file is read. Any extra variables that are present in the data file but are not relevant according to current model settings are ignored.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

B.1.5.3. Residuals

Index:	302
Scheme symbol:	`xf-rf-scaled-residuals`
C macro:	XF_RF_SCALED_RESIDUALS
Status:	optional

This section lists the values of the residuals for a particular data field variable at each iteration:

```
(302 (n residual-section-id size domain-id)
(
iteration_number unscaled_residual scaling_factor
 .
 .
 .
 ))
```

where,

n = the number of residuals

`size` = the length of the variable vector

`residual-section-id` = an integer (decimal) indicating the equation

`domain-id` = domain ID

`size` is 1 for a scalar, 2 or 3 for a vector, equal to the number of species for variables defined for each species. The `residual-section-id` indicates the equation for which the residual is stored in the section, according to the C constants defined in a header file (`xfile.h`) available in your installation area, as noted in Grid Sections (p. 2852).

The equations for which residuals are listed in the data file depend on the models active at the time the file is written. If the residual history is missing from the data file for a currently active equation, it is initialized with zeros.

B.2. Mesh Morpher/Optimizer File Formats

This section describes the format of the ASCII text files that can be used as part of the setup for the mesh morpher/optimizer, as described in Using the Mesh Morpher/Optimizer (p. 2125).

The following is an example of a file that can be read or written from the Define Control Points Dialog Box (p. 2685):

```
Control Point Positions
CP No.        X            Y             Z
1             0.73398191   -0.048181638  5.0344162
2             0.51003051   0.52833155    5.100265
3             0.10806149   0.72560567    5.1306067
4             -0.37284759  0.62890923    5.1368165
5             -0.69803506  0.21491928    5.144598
```

The following is an example of a file that can be read or written from the Motion Settings Dialog Box (p. 2749) when **Regular** is selected from the **Control Point Distribution** list in the **Regions** tab of the Mesh Morpher/Optimizer Dialog Box (p. 2734):

```
Reg     CP      Par      X       Y       Z
def-0   2       par1     -1      0       0
def-0   4       par1     2       -1      3
```

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
def-0   2       par2     0       1       0
def-1   2       par3     0       0       1
def-1   3       par1     2       0       2
```

The following is an example of a file that can be read or written from the Motion Settings Dialog Box (p. 2749) when **Unstructured** is selected from the **Control Point Distribution** list in the **Regions** tab of the Mesh Morpher/Optimizer Dialog Box (p. 2734):

```
Motion Definition
Name         Type        Par         Dir-x       Dir-y       Dir-z       Orig-x      Orig-y      Orig-z
rad-par1-1   Rad         par-1       0           0           1           0           0           0
tra-par2-1   Trans       par-2       1           1           1
rot-par3-1   Rot         par-3       1           0           0           2           1           1

Assignment to Control Points
Name         Control Points...
rad-par1-1   1           2           3           4           5           6           7
```

When creating or editing a text file used to set up the mesh morpher/optimizer, note the following:

- The file can be created from nothing using a text editor or spreadsheet program, as long as it adheres to the formats described in this section. An easier alternative is to apply some sample definitions using the appropriate dialog box, write a file, and then edit the file using a text editor or spreadsheet program.
- The file must be a tab-delimited ASCII text file.
- The names of the deformation regions (for example, `def-0`), deformation parameters (for example, `par1`), and motions (for example, `tra-par2-1`) must correspond to those that exist in the intended case file.
- When defining control points, the coordinates must lie within a deformation region in the intended case file.
- For the motion settings, the control point numbers must range between 1 and the maximum number of available control points in the intended case file.
- The direction components and coordinates can be any real numbers.

B.3. Shell Conduction Settings File Format

This section describes the format of the CSV files that can define shell conduction settings, as described in Managing Shell Conduction Walls (p. 840). The following is an example of such a file:

```
Zone ID,Zone Name,Layer Number,Material,Layer Thickness (m),Heat Generation Rate (w/m3)
15,wall-1,1,substrate,5.00E-08,0
15,wall-1,2,substrate,5.00E-08,0
15,wall-1,3,substrate,5.00E-08,0
15,wall-1,4,substrate,5.00E-08,0
15,wall-1,5,gold,2.50E-08,0
15,wall-1,6,gold,2.50E-08,0
16,wall-2,1,silicon,5.00E-08,6.25E+11
16,wall-2,2,silicon,5.00E-08,6.25E+11
16,wall-2,3,substrate,5.00E-08,0
```

When creating or editing a file that defines the shell conduction settings, note the following:

- The file can be created from nothing using a text editor or spreadsheet program, as long as it adheres to the format described in this section. An easier alternative is to apply some sample shell conduction definitions using the Shell Conduction Manager Dialog Box (p. 2806), generate a file using the **Write...** button, and then edit the file using a text editor or spreadsheet program.
- The file must be a comma-delimited ASCII file.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The first line contains the column headings shown in the previous example, and then each of the subsequent lines define the settings for a layer.
- The zone IDs, zone names, layer numbers, and materials (for example, `wall-1`, `substrate`) must correspond to those that exist in the intended case file.
- The layer thicknesses must be positive, nonzero values.
- Information will not be written for any wall that defines the heat generation rate using a UDF.

B.4. 3D Fan Curve File Format

This section describes the format of the ASCII text files that can define the relationship between pressure and flow rate for all of the 3D fan cell zones in a case file, as described in 3D Fan Zones (p. 287). The following is an example of such a file:

```
cell zone id
17 !CELL_ID
Pressure(Pa) Flowrate(m3/s)
-324.0 1.255
-215.0 1.162
-109.0 1.032
0.0 0.997
63.0 0.913
105.0 0.862
302.0 0.584
394.0 0.492
406.0 0.432
443.0 0.365
cell zone id
18 !CELL_ID
Pressure(Pa) Flowrate(m3/s)
-333.0 1.130
-215.0 1.122
-165.0 1.064
0.0 0.981
42.0 0.912
90.0 0.855
112.0 0.786
303.0 0.546
344.0 0.492
395.0 0.451
411.0 0.399
442.0 0.324
```

When creating or editing a file that defines the 3D fan zone curve, note the following:

- The file can be created from nothing using a text editor or spreadsheet program, as long as it adheres to the format described in this section.
- The file must be a tab-delimited ASCII file.
- A single file can define multiple cell zones, but it must be read in each of the applicable **Fluid** dialog boxes.
- For each section that defines the curve for a cell zone, the first three lines must follow the convention shown in the previous example, and then each of the subsequent lines must list the experimental results.
- The zone IDs (for example, 12) must correspond to those that exist in the intended case file.
- The units must match those in the case file.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Appendix C. Nomenclature

A	Area (m^2, ft^2)
$\vec{a}$	Acceleration (m/s^2, ft/s^2)
a	Local speed of sound (m/s, ft/s)
c	Concentration (mass/volume, moles/volume)
C_D	Drag coefficient, defined different ways (dimensionless)
c_p, c_v	Heat capacity at constant pressure, volume (J/kg-K, Btu/lb_m- °F)
d	Diameter; $d_p D_p$, particle diameter (m, ft)
D_H	Hydraulic diameter (m, ft)
$_{ij}D$	Mass diffusion coefficient (m^2/s, ft^2/s)
E	Total energy, activation energy (J, kJ, cal, Btu)
f	Mixture fraction (dimensionless)
$\vec{F}$	Force vector (N, lb_f)
F_D	Drag force (N, lb_f)
$\vec{g}$	Gravitational acceleration (m/s^2, ft/s^2); standard values = 9.80665 m/s^2, 32.1740 ft/s^2
Gr	Grashof number ≡ ratio of buoyancy forces to viscous forces (dimensionless)
H	Total enthalpy (energy/mass, energy/mole)
h	Heat transfer coefficient (W/m^2-K, Btu/ft^2-h- °F)
h	Species enthalpy; h^0, standard state enthalpy of formation (energy/mass, energy/mole)
I	Radiation intensity (energy per area of emitting surface per unit solid angle)
J	Mass flux; diffusion flux (kg/m^2-s, lb_m/ft^2-s)
K	Equilibrium constant = forward rate constant/backward rate constant (units vary)
k	Kinetic energy per unit mass (J/kg, Btu/lb_m)
k	Reaction rate constant, for example, k_1, k_{-1}, $k_{f,r}$, $k_{b,r}$ (units vary)
k	Thermal conductivity (W/m-K, Btu/ft-h- °F)
k_B	Boltzmann constant (1.38×10^{-23} J/molecule-K)
k, k_c	Mass transfer coefficient (units vary); also K, K_c
ℓ, l, L	Length scale (m, cm, ft, in)
Le	Lewis number ≡ ratio of thermal diffusivity to mass diffusivity (dimensionless)
m	Mass (g, kg, lb_m)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$\dot{m}$ Mass flow rate (kg/s, lb_ms)

M_w Molecular weight (kg/kmol, lb_m/lb_mmol)

M Mach number ≡ ratio of fluid velocity magnitude to local speed of sound (dimensionless)

Nu Nusselt number ≡ dimensionless heat transfer or mass transfer coefficient (dimensionless); usually a function of other dimensionless groups

p Pressure (Pa, atm, mm Hg, lb_f/ft^2)

Pe Peclet number $\equiv Re \times Pr$ for heat transfer, and $\equiv Re \times Sc$ for mass transfer (dimensionless)

Pr Prandtl number ≡ ratio of momentum diffusivity to thermal diffusivity (dimensionless)

Q Flow rate of enthalpy (W, Btu/h)

q Heat flux (W/m^2, Btu/ft^2-h)

R Gas-law constant (8.31447 $\times 10^3$ J/kmol-K, 1.98588 Btu/lb_mmol- °F)

r Radius (m, ft)

$\mathcal{R}$ Reaction rate (units vary)

Ra Rayleigh number $\equiv Gr \times Pr$; measure of the strength of buoyancy-induced flow in natural (free) convection (dimensionless)

Re Reynolds number ≡ ratio of inertial forces to viscous forces (dimensionless)

S Total entropy (J/K, J/kmol-K, Btu/lb_mmol- °F)

s Species entropy; s^0, standard state entropy (J/kmol-K, Btu/lb_mmol- °F)

Sc Schmidt number ≡ ratio of momentum diffusivity to mass diffusivity (dimensionless)

S_{ij} Mean rate-of-strain tensor (s^{-1})

T Temperature (K, °C, °R, °F)

t Time (s)

U Free-stream velocity (m/s, ft/s)

u, v, w Velocity magnitude (m/s, ft/s); also written with directional subscripts (for example, v_x, v_y, v_z, v_r)

V Volume (m^3, ft^3)

$\vec{v}$ Overall velocity vector (m/s, ft/s)

We Weber number ≡ ratio of aerodynamic forces to surface tension forces (dimensionless)

X Mole fraction (dimensionless)

Y Mass fraction (dimensionless)

α Permeability, or flux per unit pressure difference (L/m^2-h-atm, ft_3/ft^2-h-(lb_f/ft^2))

α Thermal diffusivity (m^2/s, ft^2/s)

α Volume fraction (dimensionless)

β Coefficient of thermal expansion (K_{-1})

γ Porosity (dimensionless)

γ Ratio of specific heats, c_p/c_v (dimensionless)

Δ Change in variable, final − initial (for example, Δp, Δt, ΔH, ΔS, ΔT)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

δ	Delta function (units vary)
ε	Emissivity (dimensionless)
ε	Lennard-Jones energy parameter (J/molecule)
ε	Turbulent dissipation rate (m^2/s^3, ft^2/s^3)
ε	Void fraction (dimensionless)
η	Effectiveness factor (dimensionless)
η', η''	Rate exponents for reactants, products (dimensionless)
θ_r	Radiation temperature (K)
λ	Molecular mean free path (m, nm, ft)
λ	Wavelength (m, nm, Å, ft)
μ	Dynamic viscosity (cP, Pa-s, lb_m/ft-s)
ν	Kinematic viscosity (m^2/s, ft^2/s)
ν', ν''	Stoichiometric coefficients for reactants, products (dimensionless)
ρ	Density (kg/m^3, lb_m/ft^3)
σ	Stefan-Boltzmann constant ($5.67x10^{-8}$ W/m^2-K^4)
σ	Surface tension (kg/m, dyn/cm, lb_f/ft)
σ_s	Scattering coefficient (m^{-1})
$\bar{\bar{\tau}}$	Stress tensor (Pa, lb_f/ft^2)
τ	Shear stress (Pa, lb_f/ft^2)
τ	Time scale, for example, τ_c, τ_p, τ_c (s)
τ	Tortuosity, characteristic of pore structure (dimensionless)
ϕ	Equivalence ratio (dimensionless)
ϕ	Thiele modulus (dimensionless)
Ω	Angular velocity; Ω_{ij}, Mean rate of rotation tensor (s^{-1})
ω	Specific dissipation rate (s^{-1})
Ω,Ω'	Solid angle (degrees, radians, gradians)
Ω_D	Diffusion collision integral (dimensionless)

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Appendix D. Controlling CHEMKIN-CFD Solver Parameters Using Text Commands

You have access to many parameter settings that can be used to control or improve the performance and convergence behavior of the CHEMKIN-CFD solver within ANSYS Fluent. These parameters are accessed through the ANSYS Fluent Text User Interface (TUI), through typed text commands. This appendix describes the available parameters, their default values, and their recommended use.

In general, default settings for the CHEMKIN-CFD parameters should be sufficient for most problems. However, it may be necessary to change these settings to improve performance for specific types of problems. In particular, the most appropriate solver parameters may vary depending on the nature of the chemistry set.

For example, for chemistries with many trace species that have a large impact on solution results (that is, the chemistry is "stiff"), solution tolerances should be relatively small (for example, 1.E-9 to 1.E-12 for absolute tolerances and 1.E-4 to 1.E-5 for relative tolerances). For chemistry descriptions where all of the species included are within a few orders of magnitude in terms of mass fraction, however, much looser tolerances are more appropriate. Also, depending on the characteristic time scales of the dominant chemical reactions, you may need to adjust the maximum time-step used by the CHEMKIN-CFD solver.

```
/define/models/species/CHEMKIN-CFD-parameters> basic-options
Print Level [0]
Absolute Tolerance For Gas Species [1e-08]
Relative Tolerance For Gas Species [0.0001]
Absolute Tolerance For Surface Species [1e-08]
Relative Tolerance For Surface Species [0.0001]

/define/models/species/CHEMKIN-CFD-parameters> advanced-options
Pseudo Time Stepping Solver Option (0:Basic   1:Robust) [1]
Pseudo Time Stepping Size [0.0001]
Maximum Pseudo Time Step Attempts [1]
Maximum Time Step Allowed [0.01]
Minimum Time Step Allowed [1e-10]
Initial Time Step For Solver [0]
Upper Bound For Gas Species Mass Fractions [1.01]
Lower Bound For Gas Species Mass Fractions [-1e-12]
Upper Bound For Surface Species Mass Fractions [1.1]
Lower Bound For Surface Species Mass Fractions [-0.0001]
Small Positive Value To Replace Negative Mass Fractions [1e-12]
Enter Auxiliary Input Filename [""]
Enter Arbitrary String [""]
Enter Arbitrary Value [0]
```

Table 1: Summary of Basic CHEMKIN-CFD Parameters (p. 2876) and Table 2: Summary of Advanced CHEMKIN-CFD Parameters (p. 2877) summarize basic and advanced parameters, respectively. These tables include

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

descriptions of the default values and physical units, where applicable, for each of the CHEMKIN-CFD parameters, as well as a discussion of the recommended parameter usage.

Table 1: Summary of Basic CHEMKIN-CFD Parameters

Parameter	Usage	Default Value
`Print Level`	Controls the level of diagnostic printing to the CHEMKIN-CFD diagnostic output file, where 0 indicates no printing, 1 indicates standard printing, and 2 enables a verbose level of printing that provides details about the solver operations. Note that diagnostic output can significantly contribute to compute time such that high levels of printing should be avoided unless problems are encountered that require diagnosis.	0
`Absolute Tolerance for Gas-phase Species`	Controls the smallest value of a species mass fraction that will be tracked by the CHEMKIN-CFD solver convergence algorithm. This should represent either the smallest mass fraction that is of importance in the problem or, for stiff chemistry, the smallest fraction pertaining to a species that can have a significant effect through chemical reaction on other species of interest. Typical values range from 1.E-6 to 1.E-12.	1.E-8
`Relative Tolerance for Gas-phase Species`	Controls the accuracy to which gas species mass fractions are resolved. Roughly this parameter controls the number of significant digits in the solution. However, this parameter can also be important for convergence of problems with stiff chemistry, as it is important to sufficiently resolve species that have a major impact on trace species of interest. Typical values range from 1.E-3 to 1.E-5.	1.E-4
`Absolute Tolerance for Surface Species`	Controls the smallest value of a surface species site fraction that will be tracked by the CHEMKIN-CFD solver convergence algorithm. This should represent either the smallest mass fraction that is of importance in the problem or, for stiff chemistry, the smallest fraction pertaining to a species that can have a significant effect through chemical reaction on other species of interest. Typical values range from 1.E-6 to 1.E-15.	1.E-8

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parameter	Usage	Default Value
Relative Tolerance for Surface Species	Controls the accuracy to which surface species site fractions are resolved. Roughly this parameter controls the number of significant digits in the solution. However, this parameter can also be important for convergence of problems with stiff chemistry, as it is important to sufficiently resolve species that have a major impact on trace species of interest. Typical values range from 1.E-3 to 1.E-5.	1.E-4

Table 2: Summary of Advanced CHEMKIN-CFD Parameters

Parameter	Usage	Default Value
Pseudo Time-stepping Solver Option	Determines which solver to use during pseudo time-stepping used to improve the initial guess in a cell for steady-state simulations. By default, it is set to 1 and a robust solver with strict error control will be used when the steady-state algorithm requires time-stepping assistance. You may optionally revert to a less accurate time-stepping technique, which may be computationally faster but more prone to failure, by setting it to zero. See Advanced Parameters Used in the Steady-State Solution Algorithm (p. 2879) for more information. This parameter is ignored for transient simulations.	1
Pseudo Time-stepping Size	This option is only relevant if the Pseudo Time-stepping Solver Option is set to 1 (the default setting). This controls the time interval for time-stepping when time-stepping is used to assist a steady-state calculation. See Advanced Parameters Used in the Steady-State Solution Algorithm (p. 2879) for more information. A value of 1.E-4 seconds is recommended for combustion problems.	1.E-6 [seconds]
Maximum Pseudo Time-stepping Attempts	This parameter controls how many iterations between time-stepping and steady-state solution attempts can be tried before the solver gives up and registers a failure for the cell. Usually only one attempt is needed, but the maximum can be increased from this default value in difficult cases. Note that in some cases it is most efficient to ignore a small number of cell failures early on in the simulation as long as these cells recover on subsequent iterations. See Advanced Parameters Used in the Steady-State Solution Algorithm (p. 2879) for more information. A value of 10 to 100 is recommended to minimize error messages.	1
Maximum Time Step Allowed	Maximum pseudo time-step value for time integration during steady-state solution. This value is relevant to steady-state simulations only.	1.E-2 [seconds]

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parameter	Usage	Default Value
`Minimum Time Step Allowed`	Minimum pseudo time-step value for time integration during steady-state solution. This value is relevant to steady-state simulations only.	1.E-10 [seconds]
`Initial Time Step for Solver`	Initial time-step value for use in time integration. By default, the solver will automatically choose an appropriate initial time step, which is the recommended approach. In rare cases, however, control of initial time step value may improve convergence behavior.	0 (i.e., not specified) [seconds]
`Upper Bound for Gas-species Mass Fraction`	Sets the upper bounds for the gas-phase species mass fractions during steady-state solution. In some cases, it may help convergence to allow species to go slightly over the physical bounds during interim steady-state iterations rather than forcing the bounds to be one.	1.01
`Lower Bound for Gas-species Mass Fraction`	Sets the lower bounds for the gas-phase species mass fractions during steady-state solution. In some cases, it may help convergence to allow species to go slightly negative during interim steady-state iterations rather than forcing the bounds to be zero. A value as negative as about -1.E-4 may be used.	-1.E-12
`Upper Bound for Surface Site Fraction`	Sets the upper bounds for the surface species site fractions during steady-state solution. In some cases, it may help convergence to allow species to go slightly over the physical bounds during interim steady-state iterations rather than forcing the bounds to be one.	1.1
`Lower Bound for Surface Site Fraction`	Sets the lower bounds for the surface species site fractions during steady-state solution. In some cases, it may help convergence to allow species to go slightly negative during interim steady-state iterations rather than forcing the bounds to be zero.	-1.E-4
`Small Positive Value to Replace Negative Species Fractions`	This value will be used to substitute any non-positive species fractions during interim steady-state iterations. In some cases, setting this to a very small positive number may aid convergence for the steady-state solver.	1.E-12
`Enter Auxiliary Input Filename`	Currently not used. Provided for future use.	""
`Enter Arbitrary String`	Currently not used. Provided for future use.	""
`Enter Arbitrary Value`	Currently not used. Provided for future use.	0

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

D.1. Advanced Parameters Used in the Steady-State Solution Algorithm

Many of the parameters listed in Table 2: Summary of Advanced CHEMKIN-CFD Parameters (p. 2877) are relevant only for steady-state solution. Here a brief description of the steady-state solution algorithm is provided in order to clarify the role of these parameters and their impact on solution convergence.

For steady-state problems, the ANSYS CHEMKIN-CFD solver is called by ANSYS Fluent for each cell during each Fluent iteration towards a steady state. Within each cell, the chemistry solver will attempt to determine an appropriate steady-state using a combination of a modified Newton-iteration procedure and a "pseudo" time-stepping procedure. The Newton-iteration method is used to attempt to directly iterate towards the desired solution. However, at early times in the simulation, this procedure may not succeed, especially when the initial guess is far from the actual steady solution. In such cases, the CHEMKIN-CFD module will automatically switch to a time-stepping mode to advance the initial guess closer to the physical solution. After taking a certain number of time steps, the Newton-iteration algorithm tries again. In this way, the steady-state solution is achieved with a minimum amount of time-stepping. Since time-stepping is computationally expensive, this results in a computationally efficient but robust method for achieving the steady-state solution.

You can control the number of times the CHEMKIN-CFD module is allowed to use time-stepping to advance a solution towards steady-state before reporting a failure for any individual cell (see `Maximum Pseudo Time-stepping Attempts` in Table 2: Summary of Advanced CHEMKIN-CFD Parameters (p. 2877)).

In performing the pseudo time-stepping procedure, there are two solver options - a "basic" time-stepping procedure and a "robust" time-stepping procedure (see `Pseudo Time-stepping Solver Option` in Table 2: Summary of Advanced CHEMKIN-CFD Parameters (p. 2877)). The "robust" procedure (default) uses an integration method that is very accurate with error control enforced on all solution variables. This can sometimes be more time consuming but is generally more reliable in tracking transient states towards an eventual steady condition. The "basic" solver option is much more approximate in the time-tracking technique, but is often faster and sufficiently robust for some cases.

When the "robust" time-stepping procedure is selected, you can specify the time interval used for each attempt to advance the solution using time integration (See `Pseudo Time-stepping Size` in Table 2: Summary of Advanced CHEMKIN-CFD Parameters (p. 2877)). The actual time steps used during the time integration will be automatically determined based on the rate of change of the solution variables. The time interval, however, controls the total time advanced before returning the solution to be used as the next initial guess in the Newton-iteration.method.

In addition to the pseudo time-stepping controls, you may also control certain parameters that affect the convergence behavior of the Newton-iteration algorithm itself. In particular, you can modify the solution bounds imposed during the iterations (see `Upper Bounds and Lower Bounds parameters` in Table 2: Summary of Advanced CHEMKIN-CFD Parameters (p. 2877)). By extending the bounds slightly outside of the physical limits, the Newton iteration can more easily resolve species fractions that are very near to those limits. This can be especially helpful in early iterations where variable values may be changing rapidly. It is also helpful to try to "reset" any negative species fractions to very small values between iterations, so that these values do not have unphysical effects on reaction rates. For this purpose, you can specify a small positive value, which should not exceed the specified absolute tolerance, as a replacement value for negative fractions (see `Small Positive Value to Replace Negative Species Fractions` in Table 2: Summary of Advanced CHEMKIN-CFD Parameters (p. 2877)).

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

D.2. Setting Up Monitor Cells for the ANSYS CHEMKIN-CFD Chemistry Solver

At the same level of the basic-options and advanced-options in the Text User Interface, that is, under `define/models/species/CHEMKIN-CFD-parameters`, you can add, delete, and list `monitor cells`.

By specifying a `monitor cells` in the CHEMKIN-CFD options, you are requesting additional printout diagnostics for every iteration or time-step at this cell location. Such printout can add significantly to computational times and should be avoided unless there is a problem has been encountered and more diagnosis is needed.

Important

Using monitor cells may add significantly to the job run time. It is recommended that monitor cells only be included if extra diagnosis is required to resolve a convergence problem or local mesh issue, for example.

Monitor cells are added using the `add-cell-monitor` option and providing the physical coordinates that correspond as closely as possible to the centroid location of the desired cell in the Fluent mesh. You can create more than one cell monitor and you can be delete all of them using the `delete-cell-monitor option`. You may also list the currently activated monitor cells using the `list-cell-monitor option`.

D.3. Diagnostic Files and Error Messages

In addition to the standard ANSYS Fluent files produced during a run using the CHEMKIN-CFD solver, the CHEMKIN-CFD module creates special files to hold diagnostic output and error messages that may be helpful in diagnosing a problem with the pre-processing or run steps. In addition, error messages may be written in the ANSYS Fluent Graphical User Interface window during run-time processing.

Table 3: Diagnostic Output Files Created During a CHEMKIN-CFD Run (p. 2880) lists additional files that may be created by the chemistry solver, the data printed to them, and an indication of where they are created in the ANSYS Fluent run process. All files are created within the working directory of the simulation run. The actual names of many of the files will depend on variables, such as the chemistry-set file names, the case-file name, the cell number, iteration or time-step number, and the processor id. Such variables are indicated by < > in Table 3: Diagnostic Output Files Created During a CHEMKIN-CFD Run (p. 2880).

Table 3: Diagnostic Output Files Created During a CHEMKIN-CFD Run

Filename	When File Is Created	Contents Description
`<gas_chemistry>.out`	Always created during pre-processing, upon saving the CHEMKIN mechanism definition.	This file echoes back the gas-phase chemistry reaction set, species names, and elements in formatted text. In addition, detailed error messages are contained in this file in case an error is encountered while processing the gas-phase chemistry input file and gas-phase thermodynamic data.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Filename	When File Is Created	Contents Description
chem.asc	Always created during pre-processing, upon saving the CHEMKIN mechanism definition.	A "Linking File" used to transfer the pre-processed information to the CHEMKIN-CFD module at run-time. This file is typically not viewed by the user, but may occasionally be requested by Technical Support during problem diagnosis.
<surface_chemistry>.out	Created during pre-processing, only if surface chemistry box is checked in CHEMKIN mechanism definition.	This file echoes back the surface chemistry reaction set and the surface phase and species names. In addition, detailed error messages are contained in this file in case an error is encountered while processing the surface chemistry input file and surface-species thermodynamic data.
surf.asc	Created during pre-processing, only if surface chemistry box is checked in CHEMKIN mechanism definition.	A "Linking File" used to transfer the pre-processed information to the CHEMKIN-CFD module at run-time. This file is typically not viewed by the user, but may occasionally be requested by Technical Support during problem diagnosis.
<transport_data>.out	Created during pre-processing, only if transport-property box is checked in CHEMKIN mechanism definition.	This file echoes back the transport-data fitting coefficients in formatted text. In addition, detailed error messages are contained in this file in case an error is encountered while processing the gas-phase transport data.
tran.asc	Created during pre-processing, only if transport-property box is checked in CHEMKIN mechanism definition.	A "Linking File" used to transfer the pre-processed information to the CHEMKIN-CFD module at run-time. This file is typically not viewed by the user, but may occasionally be requested by ANSYS Technical Support during problem diagnosis.
KINetics-log.txt	Created when the simulation run is initiated in the subdirectory fluentKINlog.	Messages are written to this file that record the progress of chemistry-set pre-processor and CHEMKIN-CFD solver. The printing level during the simulation run can be controlled by Print Level as indicated in Table 1: Summary of Basic CHEMKIN-CFD Parameters (p. 2876).
KIN-CELL_<cell_id>_<iteration_number>.DBG	Created for Monitor Cells (see Setting Up Monitor Cells for the ANSYS CHEMKIN-CFD Chemistry Solver (p. 2880)) at every iteration or time-step, any time that the Print Level (Table 1: Summary of	Includes debug information, including the actual input sent into the CHEMKIN-CFD module by ANSYS Fluent for the particular cell and iteration. This information may be requested by ANSYS Technical Support when diagnosing cell failures.

Filename	When File Is Created	Contents Description
	Basic CHEMKIN-CFD Parameters (p. 2876)) is >0. Also created any time that a cell fails in the CHEMKIN-CFD solver and the `Print Level` is >0.	

D.4. Error Messages Printed in the ANSYS Fluent Graphical User Interface

Advanced Parameters Used in the Steady-State Solution Algorithm (p. 2879) lists possible error messages that may appear in the ANSYS Fluent Graphical User Interface during run using the CHEMKIN-CFD solver. The table includes recommended actions that may help resolve or narrow down the problem. In some cases, you may be referred to some of the additional diagnostic output files generated by the CHEMKIN-CFD module and listed in Table 3: Diagnostic Output Files Created During a CHEMKIN-CFD Run (p. 2880).

Table 4: Error Messages that May Be Printed to the Fluent GUI

Setting	Value
`Cannot read gas-phase chemistry input file.`	Verify the filename and file path for the gas-phase chemistry input file in the **CHEMKIN Mechanism Import** dialog box and re-import.
`Cannot read surface chemistry input file.`	Verify the filename and file path for the surface chemistry input file in the **CHEMKIN Mechanism Import** dialog box and re-import.
`Cannot read thermodynamic data file.`	Verify the filename and file path for the thermodynamic data file in the **CHEMKIN Mechanism Import** dialog box and re-import.
`Cannot read transport properties data file.`	Verify the filename and file path for the transport-property data file in the **CHEMKIN Mechanism Import** dialog box and re-import.
`Cannot read gas-phase chemistry linking data file during gas-phase pre-processing.`	Check the `<gas_chemistry>.out` file in the working directory for error messages.
`Cannot read gas-phase chemistry linking data file during surface pre-processing.`	Check the `<gas_chemistry>.out` file in the working directory for error messages.
`Cannot read gas-phase chemistry linking data file during transport pre-processing.`	Check the `<gas_chemistry>.out` file in the working directory for error messages.
`Cannot read surface chemistry linking data file during surface pre-processing.`	Check the `<surface_chemistry>.out` file in the working directory for error messages.
`Cannot read transport properties linking data file during transport pre-processing.`	Check the `<transport_data>.out` file in the working directory for error messages.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
Cannot write gas-phase chemistry output file.	Check file permissions on working directory and make sure that there is only one CHEMKIN-CFD process using these files at a time.
Cannot write surface chemistry output file.	Check file permissions on working directory and make sure that there is only one CHEMKIN-CFD process using these files at a time.
Cannot write transport properties output file.	Check file permissions on working directory and make sure that there is only one CHEMKIN-CFD process using these files at a time.
Cannot write gas-phase chemistry linking data file.	Check file permissions on working directory and make sure that there is only one CHEMKIN-CFD process using these files at a time.
Cannot write surface chemistry linking data file.	Check file permissions on working directory and make sure that there is only one CHEMKIN-CFD process using these files at a time.
Cannot write transport properties linking data file.	Check file permissions on working directory and make sure that there is only one CHEMKIN-CFD process using these files at a time.
Gas-phase chemistry file is NULL.	Verify gas-phase chemistry file path in the CHEMKIN Mechanism definition and re-import the chemistry files.
Surface chemistry file is NULL.	Verify surface-chemistry file path in the CHEMKIN Mechanism definition and re-import the chemistry files.
Thermodynamic data file is NULL.	Verify thermodynamic-data file path in the CHEMKIN Mechanism definition and re-import the chemistry files
Transport properties data file is NULL.	Verify transport-data file path in the CHEMKIN Mechanism definition and re-import the chemistry files.
Failure in pre-processing of gas-phase chemistry mechanism.	Check the <gas_chemistry>.out file in the working directory for error messages.
Failure in pre-processing of surface chemistry mechanism.	Check the <surface_chemistry>.out file in the working directory for error messages.
Failure in pre-processing of transport properties.	Check the <transport_data>.out file in the working directory for error messages.
Failure in CHEMKIN Gas-phase library call.	Check to see if the ANSYS Fluent solution has diverged or has unphysical values for the variables. Try turning Print Level > 0 and

Setting	Value
	setting a monitor cell to obtain more information.
`Failure in CHEMKIN Surface library call.`	Check to see if the ANSYS Fluent solution has diverged or has unphysical values for the variables. Try turning `Print Level` > 0 and setting a monitor cell to obtain more information.
`Failure in CHEMKIN Transport library call.`	Check to see if the ANSYS Fluent solution has diverged or has unphysical values for the variables. Try turning `Print Level` > 0 and setting a monitor cell to obtain more information.
`Need to run CHEMKIN-CFD PrePro-cess first.`	Re-import the CHEMKIN chemistry set and make sure it is selected in the **Species Model** dialog box (under **Setup**/**Models**).
`No surface data for the chemistry set.`	Either de-select the Surface Chemistry option in the **CHEMKIN Mechanism Import** dialog box, or make sure that a valid surface chemistry file is indicated.
`No transport data for the chem-istry set.`	Either de-select the Transport Data option in the **CHEMKIN Mechanism Import** dialog box, or make sure that a valid transport data file is indicated.
`Failure in writing data into the diagnostics file.`	Check file permissions on the working directory and make sure that there is only one CHEMKIN-CFD process using these files at a time.
`Error in CHEMKIN-CFD model, see diagnostic output file for de-tails.`	Check the `KINetics-log.txt` file in the subdirectory `fluentKINlog` for error messages. See Error Messages Printed in the ANSYS Fluent Graphical User Interface (p. 2882) for more information.

D.5. Diagnostic Messages in the `KINetics-log.txt` File

The `KINetics-log.txt` file is a "catch-all" for many diagnostic messages that are generated by the CHEMKIN-CFD module. The most common types of messages are error messages related to setup or cell convergence.

Cell-convergence error messages can take a variety of forms. The most common are summarized in Table 5: Other Error Messages in `KINetics-log.txt` (p. 2885). In general, if there are just a few failed cells at the beginning of a simulation it is safe to ignore them. When the number of failed cells continues to grow with each iteration, there is likely an issue with the problem set-up or with the chemistry data.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Table 5: Other Error Messages in KINetics-log.txt (p. 2885) contains some broad suggestions of ways to improve cell-convergence behavior based on the type of cell failure message received.

Table 5: Other Error Messages in KINetics-log.txt

Error Message	Recommended Action
FAILURE INITIALIZING SETTINGS FOR THE KINetics STEADY-STATE GAS-PHASE CHEMISTRY SOLVER	Most likely suggests an error in setting the convergence criteria (that is absolute and relative tolerance) and the pseudo-steady-state time stepping parameters. Check that the user-provided values are reasonable.
WARNING: Exceeded allowed time-stepping intervals in cell	Indicates that a convergence problem was encountered during steady-state solution, where time-stepping is used to help advance the steady-state iterations. In this case, you can either ignore the message (see if the cell convergence recovers in subsequent iterations) or try to increase either the time interval or the number of allowed time-stepping intervals (See Table 2: Summary of Advanced CHEMKIN-CFD Parameters (p. 2877) for more information).
KINETICS ERROR: SOLUTION FAILED	Indicates that the cell failed to converge (either steady-state or transient).In this case, you can either ignore the message (see if the cell convergence recovers in subsequent iterations) or try to modify some of the user-controlled parameters to improve convergence (See Table 2: Summary of Advanced CHEMKIN-CFD Parameters (p. 2877) for more information).
Warning - The transient chemistry solver tried 500 steps without convergence in this cell; 500 more steps will be allowed	Indicates that the problem is difficult to converge so that the transient solver is taking extra steps. However, it is only a warning and the cell has not actually failed. If you receive an excessive number of these messages, try reducing the absolute or relative solution tolerances that control the CHEMKIN-CFD solver.
KINETICS ERROR: Problem getting species names; Check that the CHEMKIN files have been processed correctly	Indicates a possible error in the pre-processing of the gas-phase chemistry input file. Check the <gas_chemistry>.out file for error messages.
KINETICS ERROR: Problem getting surface chemistry info; Check that the CHEMKIN files have been processed correctly	Indicates a possible error in the pre-processing of the surface chemistry input file. Check the <surface_chemistry>.out file for error messages.
DASPK:...	These messages are from the CHEMKIN-CFD transient solver, DASPK. The exact message

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Error Message	Recommended Action
	usually contains more information about the problem.

Note

During steady-state simulations, once an iteration occurs where there are no failed cells, the effect of previously failed cells in earlier iterations is completely erased.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Bibliography

[1] *CRANIUM, Version 1.0*. Molecular Knowledge Systems, Inc., Bedford, NH. 2007.

[2] *FIELDVIEW Reference Manual, Software Release Version 10*. Intelligent Light. 2004.

[3] *KINetics for Fluent, Version 2.4*. Reaction Design, Inc., San Diego, CA. 2009.

[4] *VdmTools Programmer Manual, Version 3.2.0*. Visual Kinematics, Inc. 2005.

[5] T. Ahmad, S. L. Plee, and J. P. Myers. *"Computation of Nitric Oxide and Soot Emissions from Turbulent Diffusion Flames"*. *J. of Engineering for Gas Turbines and Power*. 107. 48–53. 1985.

[6] A. A. Amsden, P. J. O'Rourke, and T. D. Butler. *"KIVA-2: A Computer Program for Chemically Reactive Flows with Sprays"*. *Technical Report LA-11560-MS, UC-96*. Los Alamos National Laboratory, Los Alamos, New Mexico. May 1989.

[7] M. J. Assael, J. P. M. Trusler, and T. F. Tsolakis. *"Thermophysical Properties of Fluids"*. *Technical report*. Imperial College Press, London. 1996.

[8] R. H. Aungier. *"A Fast, Accurate Real Gas Equation of State for Fluid Dynamic Analysis Applications"*. *Journal of Fluids Engineering*. 117. 277–281. 1995.

[9] R. Barrett, M. Berry, T. F. Chan, J. Demmel, J. Donato, J. Dongarra, V. Eijkhout, R. Pozo, C. Romine, and H. Van der Vorst. *"Templates for the Solution of Linear Systems: Building Blocks for Iterative Methods"*. *SIAM, 2nd edition*. Philadelphia, Pennsylvania. 1994.

[10] A. Bejan. *Convection Heat Transfer*. John Wiley and Sons, New York. 1984.

[11] D. T. Blackstock. *Fundamentals of Physical Acoustics*. John Wiley & Sons. 2000.

[12] C. T. Bowman. *"Chemistry of Gaseous Pollutant Formation and Destruction"*. *Fossil Fuel Combustion*. W. Bartok and A. F. Sarofim, editors. J. Wiley and Sons, Canada. 1991.

[13] R. N. Bracewell. *The Fourier transform and its applications*. McGraw Hill. 1986.

[14] W.-P. Breugem. *"The Effective Viscosity of a Channel-type Porous Medium"*. *Phys. of Fluids*. 19. 10. 2007.

[15] H. Brinkman. *"A Calculation of the Viscous Force Exerted by a Flowing Fluid on a Dense Swarm of Particles"*. *Applied Scientific Research*. A1. 27–34.

[16] H. Brinkman. *"The Viscosity of Concentrated Suspensions and Solutions"*. *J. Chem. Phys*. 20. 4. 571–584. 1952.

[17] R. J. Brooks and A. T. Corey. *"Hydraulic properties of porous media"*. *Hydrology Paper 3*. Colorado State University. Fort Collins, Colorado, USA. 1964.

[18] S. J. Brookes and J. B. Moss. *"Prediction of Soot and Thermal Radiation in Confined Turbulent Jet Diffusion Flames"*. *Combustion and Flame*. 116. 486–503. 1999.

[19] J. R. Cash and A. H. Karp. *"A variable order Runge-Kutta method for initial value problems with rapidly varying right-hand sides"*. *ACM Transactions on Mathematical Software*. 16. 201–222. 1990.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[20] T. Cebeci and P. Bradshaw. *Momentum Transfer in Boundary Layers*. Hemisphere Publishing Corporation, New York. 1977.

[21] P. J. Coelho and M. G. Carvalho. *"Modeling of Soot Formation and Oxidation in Turbulent Diffusion Flames"*. *J. of Thermophysics and Heat Transfer*. 9(4). 644–652. 1995.

[22] M. F. Cohen and D. P. Greenberg. *"The Hemi-Cube: A Radiosity Solution for Complex Environments"*. *Computer Graphics*. 19(3). 31–40. 1985.

[23] H. W. Coleman, B. K. Hodge, and R. P. Taylor. *"A Re-Evaluation of Schlichting's Surface Roughness Experiment"*. *Journal of Fluids Engineering*. 106. 1984.

[24] E. H. Cuthill and J. McKee. *"Reducing Bandwidth of Sparse Symmetric Matrices"*. *In Proc. ACM 24th National Conf. New York*. 157–172. 1969.

[25] J. E. Dec. *"A Conceptual Model of DI Diesel Combustion Based on Laser Sheet Imaging"*. *SAE Technical Paper 970873*. SAE. 1997.

[26] J. Ding and D. Gidaspow. *"A Bubbling Fluidization Model Using Kinetic Theory of Granular Flow"*. *AIChE J*. 36(4). 523–538. 1990.

[27] Y. Egorov and F. Menter. *"Experimental Implementation of the RPI Wall Boiling Model in CFX-5.6"*. *Technical Report*. ANSYS/TR-04-10, ANSYS GmbH. 2004.

[28] S. Ergun. *"Fluid Flow through Packed Columns"*. *Chem. Eng. Prog*. 48(2). 89–94. 1952.

[29] G. M. Faeth. *"Spray Atomization and Combustion"*. *Technical Report 1986-136*. AIAA. 1986.

[30] J. E. Ffowcs-Williams and D. L. Hawkings. *"Sound Generation by Turbulence and Surfaces in Arbitrary Motion"*. *Proc. Roy. Soc. London*. A264. 321–342. 1969.

[31] E. U. Finlayson, D. K. Arasteh, C. Huizenga, M. D. Rubin, and M. S. Reilly. *"WINDOW 4.0: Documentation of Calculation Procedures"*. *Technical Report Publication LBL-33943/ TA-309*. Lawrence Berkeley Laboratory, Energy and Environmental Division, Berkeley, CA 94720. 1993.

[32] K.–Y. Fung and H. Ju. *"Broadband Time-Domain Impedance Models"*. *AIAA Journal*. 39(8). January 2001.

[33] K.–Y. Fung, H. Ju, and B. Tallapragada. *"Impedance and Its Time-Domain Extensions"*. *AIAA Journal*. 38(1). 30–38. January 2000.

[34] A. Ghobadian and S. A. Vasquez. *"A General Purpose Implicit Coupled Algorithm for the Solution of Eulerian Multiphase Transport Equation"*. *International Conference on Multiphase Flow*. Leipzig, Germany. 2007.

[35] N. E. Gibbs, W. G. Poole, Jr., and P. K. Stockmeyer. *"An algorithm for reducing the bandwidth and profile of a sparse matrix"*. *SIAM J. Numer. Anal*. 13. 236–250. 1976.

[36] M. B. Giles. *"Non-Reflecting Boundary Conditions for the Euler Equations"*. *Technical Report 88-1*. Computational Fluid Dynamics Laboratory, Massachusetts Institute of Technology, Cambridge, MA. 1988.

[37] R.A. Gore and C.T. Crowe. *"Effect of particle size on modulating turbulent intensity"*. *Int. J. Multiphase Flow*. 15. 279–285. 1989.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[38] J. Gottgens, F. Mauss, and N. Peters. *"Analytic Approximations of Burning Velocities and Flame Thicknesses of Lean Hydrogen, Methane, Ethylene, Ethane, Acetylene and Propane Flames"*. *In Twenty-Fourth Symposium (Int.) on Combustion, Pittsburgh*. 129–135. 1992.

[39] P. M. Gresho, R. L. Lee, and R. L. Sani. *"On the Time-Dependent Solution of the Incompressible Navier-Stokes Equations in Two and Three Dimensions"*. *In Recent Advances in Numerical Methods in Fluids*. Pineridge Press, Swansea, U. K. 1980.

[40] R. J. Hall, M. D. Smooke, and M. B. Colket. *Physical and Chemical Aspects of Combustion*. Gordon and Breach. 1997.

[41] R. A. W. M. Henkes and C. J. Hoogendoorn. *"Scaling of the Turbulent Natural Convection Flow in a Heated Square Cavity"*. *Trans. of the ASME*. 116. 400–408. May 1994.

[42] J. B. Heywood. *Internal Combustion Engine Fundamentals*. McGraw-Hill, New York, rev. edition. 1988.

[43] C. Hirsch. *Numerical Computation of Internal and External Flows: Computational Methods for Inviscid and Viscous Flows, volume 2*. John Wiley & Sons, New York. 1984.

[44] J. O. Hirschfelder, C. F. Curtiss, and R. B. Bird. *Molecular Theory of Gases and Liquids*. John Wiley & Sons, New York. 1954.

[45] K. Hsu and A. Jemcov. *"Numerical investigation of detonation in premixed hydrogen-air mixture - assessment of simplified chemical mechanisms"*. *Technical Report AIAA-2000-2478*. American Institute of Aeronautics and Astronautics, Fluids 2000 Conference and Exhibit, Denver, CO. June 2000.

[46] G. W. Jackson and D. F. James. *"The Permeability of Fibrous Porous Media"*. *Canadian Journal of Chemical Engineering*. 64(3). 364–374. June 1986.

[47] A. Jemcov and J. P. Maruszewski. *"Shape Optimization Based on Downhill Simplex Optimizer and Free-Form Deformation in General Purpose CFD Code"*. *17th Annual Conference of CFD Society of Canada*. Ottawa, Ontario, Canada. 2009.

[48] P. C. Johnson and R. Jackson. *"Frictional-Collisional Constitutive Relations for Granular Materials, with Application to Plane Shearing"*. *J. Fluid Mech*. 176. 67–93. 1987.

[49] G. Karypis and V. Kumar. *METIS - A Software Package for Partitioning Unstructured Graphs, Partitioning Meshes, and Computing Fill-Reducing Orderings of Sparse Matrices, Version 3.0. Manual*. University of Minnesota and Army HPC Research Center. 1997.

[50] A. Kaufmann, F. Nicoud, and T. Poinsot. *Flow Forcing Techniques for Numerical Simulation of Combustion Instabilities*. *Combustion and Flame*. 131. 371–385. 2002.

[51] R. J. Kee, F. M. Rupley, and J. A. Miller. *"SURFACE CHEMKIN: A Software Package for the Analysis of Heterogeneous Chemical Kinetics at a Solid-Surface - Gas-Phase Interface"*. *Technical Report SAND 96-8217*. Sandia National Labs. 2000.

[52] R. J. Kee, F. M. Rupley, J. A. Miller, M. E. Coltrin, J. F. Grcar, E. Meeks, H. K. Moffat, A. E. Lutz, G. Dixon-Lewis, M. D. Smooke, J. Warnatz, G. H. Evans, R. S. Larson, R. E. Mitchell, L. R. Petzold, W. C. Reynolds, M. Caracotsios, W. E. Stewart, P. Glarborg, C. Wang, O. Adigun, W. G. Houf, C. P. Chou, S. F. Miller, P. Ho, and D. J. Young. *CHEMKIN v. 4.0. Technical Report*. San Diego, CA. Reaction Design, Inc. 2004.

[53] T. G. Kolda, R. M. Lewis, and V. Torczon. *"Optimization by Direct Search: New Perspectives on Some Classical and Modern Methods"*. *SIAM Review*. 45. 3. 385–482. 2003.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[54] B. Koncar, I. Kljenak, and R. Mavko. *"Modeling of Local Two-Phase Flow Parameters in Upward Subcooled Flow Boiling at Low Pressure"*. *International Journal of Heat and Mass Transfer*. 47. 1499–1513. 2004.

[55] K. K. Y. Kuo. *Principles of Combustion*. John Wiley and Sons, New York. 1986.

[56] B. E. Launder and D. B. Spalding. *"The Numerical Computation of Turbulent Flow"*. *Computer Methods in Applied Mechanics and Engineering*. 3. 269–289. 1974.

[57] J. Li, Z. Zhao, et al. *"On Sound Generated Aerodynamically"*. *Proc. Roy. Soc. London*. A211. 564–587. 1952.

[58] M. J. Lighthill. *"A Comprehensive Kinetic Mechanism for CO, CH2O, and CH3OH Combustion"*. *Int. J. Chem. Kinet*. 39. 109-136. 2007.

[59] P. Lin. *Numerical Modeling of Water Waves*. Taylor & Francis. 2008.

[60] R. P. Lindsted, F. C. Lockwood, and M. A. Selim. *"Rate Constants Based on Partial Equilibrium Assumptions"*. *Internal Report No. TF/95/3, Technical report*. Imperial College, London, UK. 1995.

[61] M. S. Liou. *"A sequel to AUSM: AUSM+"*. *Journal of Computational Physics*. 129. 364–382. 1996.

[62] F. C. Lockwood, S. M. A. Rizvi, and N. G. Shah. *"Comparative Predictive Experience of Coal Firing"*. *In Proceedings Inst. Mechanical Engineers*. 200. 79–87. 1986.

[63] J. Y. Luo, R. I. Issa, and A. D. Gosman. *"Prediction of Impeller-Induced Flows in Mixing Vessels Using Multiple Frames of Reference"*. *In I ChemE Symposium Series*. 136. 549–556. 1994.

[64] B. F. Magnussen and B. H. Hjertager. *"On mathematical models of turbulent combustion with special emphasis on soot formation and combustion"*. *In 16th Symp. (Int'l.) on Combustion*. The Combustion Institute. 1976.

[65] M. Manninen, V. Taivassalo, and S. Kallio. *"On the mixture model for multiphase flow"*. *VTT Publications 288*. Technical Research Centre of Finland. 1996.

[66] B. J. McBride, S. Gordon, and M. A. Reno. *"Coefficients for Calculating Thermodynamic and Transport Properties of Individual Species"*. *NASA TM-4513*. October 1993.

[67] H. A. McGee. *Molecular Engineering*. McGraw-Hill, New York. 1991.

[68] K. I. M. McKinnon. *"Convergence of the Nelder-Mead Simplex Method a Nonstationary Point "*. *SIAM J. Optimization*. 9. 1. 148–158. 1998.

[69] F. R. Menter and Y. Egorov. *"Re-visiting the Turbulent Scale Equation"*. *Proc. IUTAM Symp. One Hundred Years of Boundary Layer Research, Göttingen*. Springer. 2004.

[70] F. R. Menter and Y. Egorov. *"A Scale-Adaptive Simulation Model Using Two-Equation Models"*. *AIAA Paper-2005-1095*. Reno NV. 2005.

[71] H. J. Merk. *"The Macroscopic Equations for Simultaneous Heat and Mass Transfer in Isotropic, Continuous and Closed Systems"*. *Appl. Sci. Res*. 8. 73–99. 1958.

[72] R. Merz, J. Kruckels, J. Mayer, and H. Stetter. *"Computation of Three Dimensional Viscous Transonic Turbine Stage Flow Including Tip Clearance Effects"*. *ASME 95-GT-76*. 1995.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[73] C. V. Naik, K. V. Puduppakkam, et al. *"Modeling Combustion Characteristics of Components of Model-Biodiesel at High Temperatures"*. *WSS/CI Spring 2008 Meeting*. Los Angeles, CA. 2008.

[74] J. A. Nelder and R. Mead. *"A Simplex Method for Function Minimization"*. *Computer Journal*. 7. 4. 308–313. 1965.

[75] Y.I. Oka et al.. *"Practical estimation of erosion damage caused by solid particle impact, Part 1: Effects of impact parameters on a predictive equation"*. *15th International Conference on Wear of Materials*. 259. 1-6. 95–101. 2005.

[76] P. J. O'Rourke. *"Collective Drop Effects on Vaporizing Liquid Sprays"*. *PhD thesis*. Princeton University, Princeton, New Jersey. 1981.

[77] M. Østberg, P. Glarborg, A. Jensen, J. E. Johnsson, L. S. Pedersen, and K. Dam-Johansen. *"A model of the coal reburning process"*. *In Proc. Combust Inst*. 27. 3027–3035. 1998.

[78] I. Owczarek and K. Blazej. *"Recommended Critical Temperatures. Part I. Aliphatic Hydrocarbons"*. *J. Phys. Chem*. 32. 1411–1427. 2003.

[79] I. Owczarek and K. Blazej. *"Recommended Critical Pressures. Part I. Aliphatic Hydrocarbons"*. *J. Phys. Chem*. 35. 1461–1474. 2006.

[80] S. V. Patankar. *Numerical Heat Transfer and Fluid Flow*. Hemisphere, Washington, DC. 1980.

[81] S. V. Patankar, C. H. Liu, and E. M. Sparrow. *"Fully Developed Flow and Heat Transfer in Ducts Having Streamwise-Periodic Variations of Cross-Sectional Area"*. *ASME J. of Heat Transfer*. 99. 180–186. 1977.

[82] D. Y. Peng and D. B. Robinson. *"A New Two-Constant Equation of State"*. *Industrial and Engineering Chemistry: Fundamentals*. 15. 59–64. 1976.

[83] R. H. Perry, D. W. Gree, and J. O. Maloney. *Perry's Chemical Engineers' Handbook*. McGraw-Hill, New York, 6th edition. 1984.

[84] A. A. F. Peters and R. Weber. *"Mathematical Modeling of a 2.25 MWt Swirling Natural Gas Flame. Part 1: Eddy Break-up Concept for Turbulent Combustion; Probability Density Function Approach for Nitric Oxide Formation"*. *Combustion Science and Technology*. 110. 67–101. 1995.

[85] T. J. Poinsot and S. K. Lele. *"Boundary Conditions for Direct Simulation of Compressible Viscous Flows"*. *Journal of Computational Physics*. 101. 104–129. 1992.

[86] T. J. Poinsot and L. Selle. *"Actual Impedance of Nonreflecting Boundary Conditions: Implications for Computation of Resonators"*. *AIAA Journal*. 42(5). 958–964. May 2004.

[87] B. E. Poling, J. M. Prausnitz, and J. P. O'Connell. *The properties of Gases and Liquids*. McGraw-Hill, International Edition, 5th edition. 2007.

[88] S. B. Pope. *ISAT-CK (Version 3.0) User's Guide and Reference Manual*. 2000.

[89] M. J. D. Powell. *"The NEWUOA software for unconstrained optimization without derivatives"*. *Large-Scale Nonlinear Optimization (Nonconvex Optimization and Its Applications)*. 83. 255–297. G. Di Pillo and M. Roma, editors. Springer. 2006.

[90] W. H. Press, S. A. Teukolsky, W. T. Vetterling, and B. P. Flannery. *Numerical Recipes: The Art of Scientific Computing, Third Edition*. Cambridge University Press, Cambridge, England. 2007.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[91] C. L. Rasmussen and A. E. Rasmussen. *"Sensitizing Effects of NOx on CH4 Oxidation at High Pressure"*. *Comb. Flame*. 154. 233. 1949.

[92] O. Redlich and J. N. S. Kwong. *"On the Thermodynamics of Solutions. An Equation of State. Fugacities of Gaseous Solutions"*. *Chem. Rev*. 44. 3. 529-545. 2008.

[93] W. C. Reynolds. *Thermodynamic Properties in SI: Graphs, Tables, and Computational Equations for 40 Substances*. Department of Mechanical Engineering, Stanford University. 1979.

[94] J. W. Rose and J. R. Cooper. *Technical Data on Fuel*. Scottish Academic Press, Edinburgh. 1977.

[95] J. W. Rose and J. R. Cooper, editors. *Technical Data on Fuels*. Wiley, 7th edition. 1977.

[96] H. H. Rosenbrock. *"An Automatic Method for Finding the Greatest or Least Value of a Function"*. *Computer Journal*. 3. 3. 175–184. 1960.

[97] A. Saxer. *"A Numerical Analysis of a 3D Inviscid Stator/Rotor Interaction Using Non-Reflecting Boundary Conditions"*. *PhD thesis*. Massachusetts Institute of Technology, Cambridge, Massachusetts. March 1992.

[98] S. Sarkar and L. Balakrishnan. *"Application of a Reynolds-Stress Turbulence Model to the Compressible Shear Layer"*. ICASE Report 90-18*NASA CR 182002*. 1990.

[99] P. R. Spalart, S. Deck, M. L. Shur, K. D. Squires, M. Kh. Strelets, and A. Travin. *"A New Version of Detached-eddy Simulation, Resistant to Ambiguous Grid Densities "*. *Theoretical and Computational Fluid Dynamics*. Springer Berlin / Heidelberg. Volume 20, Number 3. July, 2006.

[100] H. Schlichting. *Boundary-Layer Theory*. McGraw-Hill, New York. 1979.

[101] H. Schlichting and K. Gersten. *Grenzschicht-Theorie*. Springer-Verlag, Berlin, Heidelberg, New York. 1997.

[102] A. A. Shabana. *Computational Dynamics*. John Wiley and Sons, New York. 1994.

[103] J. Shin and T. H. Bond. *"Experimental and Computational Ice Shapes and Resulting Drag Increase for a NACA 0012 Airfoil"*. *NASA Technical Memorandum 105743*. 1992.

[104] G. M. Shroff and H. B. Keller. *"Stabilization of unstable procedures: The recursive projection method"*. *SIAM J. Numer. Anal*. 30(4). 1099–1120. 1993.

[105] M. L. Shur, P. R. Spalart, M. K. Strelets, and A. K. Travin. *"A Hybrid RANS-LES Approach With Delayed-DES and Wall-Modelled LES Capabilities"*. *International Journal of Heat and Fluid Flow*. 29:6. December 2008. 1638-1649.

[106] Y. R. Sivathanu and G. M. Faeth. *"Generalized State Relationships for Scalar Properties in Non-Premixed Hydrocarbon/Air Flames"*. *Combustion and Flame*. 82. 211–230. 1990.

[107] I. B. Sladkov. *"Physicochemical Properties of Methyl- and Ethylhalosilanes"*. *Russian Journal of Applied Chemistry*. 74. 1801–1805. 2001.

[108] A. L. Smith. *"Family Plots for Evaluating Physical Properties of Organosilicon Compounds"*. *AIChE Journal*. 40. 373–377. 1994.

[109] G. P. Smith, D. M. Golden, et al. *"GRI-Mech 3.0"*. 2008. *http://www.me.berkeley.edu/gri_mech*.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[110] P. L. Smith, Jr. and M. Van Winkle. *"Discharge coefficients through perforated plates at reynolds numbers of 400 to 3,000"*. *AIChE Journal*. 4(3). 266–268. 1958.

[111] D. O. Snyder, E. K. Koutsavdis, and J. S. R. Anttonen. *"Transonic store separation using unstructured cfd with dynamic meshing"*. *Technical Report AIAA-2003-3913*. American Institute of Aeronautics and Astronautics, 33th AIAA Fluid Dynamics Conference and Exhibit. 2003.

[112] G. Soave. *"Equilibrium Constants from a modified Redlich-Kwong equation of State"*. *Chemical Engineering Science*. 27. 1197. 1972.

[113] G. R. Somayajulu. *"Estimation Procedures for Critical Constants"*. *J. Chem. Eng. Data*. 34. 106–120. 1989.

[114] K. Stueben. *"Introduction to Algebraic Multigrid"*. *Multigrid*. C. W. Oosterlee, U. Trottenberg, and A. Schuller, editors. Academic Press, New York. 413–532. 2001.

[115] K. Sutton and P. A. Gnoffo. *"Multi-component Diffusion with Application to Computational Aerothermodynamics"*. *AIAA Paper 98-2575*. AIAA. 1998.

[116] M. Syamlal, W. Rogers, and T. J. O'Brien. *MFIX Documentation: Volume 1, Theory Guide*. DOE/METC-9411004, NTIS/DE9400087 National Technical Information Service, Springfield, VA. 1993.

[117] L. Talbot et al. *"Thermophoresis of Particles in a Heated Boundary Layer"*. *J. Fluid Mech*. 101(4). 737–758. 1980.

[118] W. Tang and K. Brezinsky. *"Chemical kinetic simulations behind reflected shock waves"*. *International Journal of Chemical Kinetics*. 38(2). 75–97. February 2006.

[119] R. I. Tanner. *Engineering Rheology*. Clarendon Press, Oxford, rev. edition. 1988.

[120] R. Taylor and R. Krishna. *Multicomponent Mass Transfer*. Wiley, New York. 1993.

[121] C. Temperton. *"Implementation of a Self-Sorting In-Place Prime Factor FFT Algorithm"*. *Journal of Computational Physics*. 58. 283–299. 1985.

[122] P. A. Tesner, T. D. Snegiriova, and V. G. Knorre. *"Kinetics of Dispersed Carbon Formation"*. *Combustion and Flame*. 17. 253–260. 1971.

[123] K. W. Thompson. *"Time Dependent Boundary Conditions for Hyperbolic Systems"*. *Journal of Computational Physics*. 68. 1–24. 1987.

[124] K. W. Thompson. *"Time Dependent Boundary Conditions for Hyperbolic Systems II"*. *Journal of Computational Physics*. 89. 439–461. 1990.

[125] V. Torczon. *"On the Convergence of the Multidirectional Search Algorithm"*. *SIAM J. Optimization*. 1. 1. 123–145. 1991.

[126] H. Wagner. *"Soot Formation in Combustion"*. *In 17th Symp. (Int'l.) on Combustion*. The Combustion Institute. 3–19. 1979.

[127] J. M. Weiss. *"Calculations of Reacting Flowfield Involving Stiff Chemical Kinetics"*. *AIAA-99-3369*. January 1999.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[128] J. M. Weiss, J. P. Maruszewski, and W. A. Smith. *"Implicit Solution of the Navier-Stokes Equations on Unstructured Meshes"*. *Technical Report AIAA-97-2103, 13th AIAA CFD Conference*. Snowmass, CO. July 1997.

[129] C. Westbrook and F. Dryer. *"Simplified Reaction Mechanisms for Oxidation of Hydrocarbon Fuels in Flames"*. *Comb. Sci. Tech*. 27. 31–43. 1981.

[130] D. C. Wilcox. *Turbulence Modeling for CFD*. DCW Industries, Inc., La Canada, California. 1998.

[131] V. Zimont, W. Polifke, M. Bettelini, and W. Weisenstein. *"An Efficient Computational Model for Premixed Turbulent Combustion at High Reynolds Numbers Based on a Turbulent Flame Speed Closure"*. *J. of Gas Turbines Power*. 120. 526–532. 1998.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Index

Symbols

.fluent files, 108
1D Simulation Library dialog box, 2662
2.5D model, 686
2.5D surface remeshing method, 685
2D fan boundary, 389
  user-defined, 427
3D fan zone, 287
  file format, 2869
  inputs for, 289
  theory, 287

A

ABAQUS files
  exporting, 69
    after a calculation, 71
    during a transient calculation, 86
  importing, 62
  mapping data with, 97
absolute angular coordinate, definition, 1957
absolute pressure, 531
absolute pressure, definition, 1958
absolute reference frame, 1263
absolute velocity, 608, 611, 616, 619, 624, 639, 1592, 1594, 1598
absolute velocity formulation, 609
absorbed IR solar flux, definition, 1958
absorbed radiation flux, definition, 1958
absorbed visible solar flux, definition, 1958
absorption coefficient, 516, 868
  composition-dependent, 516
  constant, 516
  effect of particles on, 517
  effect of soot on, 517, 1186, 1188, 1193
  non-gray radiation, 517
  WSGGM, 516
absorption coefficient, definition, 1958
accretion, 1251, 1309, 1965
  reporting, 1360
accuracy, 133
  first-order, 1555
  first-to-higher order, 1555
  second-order, 1555
acentric factor, definition, 1958
acoustic power level, definition, 1959
acoustic power, definition, 1958
Acoustic Receivers dialog box, 1217, 2293
acoustic signals, 1206
Acoustic Signals dialog box, 1220, 2547
Acoustic Sources dialog box, 1214, 2292
Acoustic Sources FFT dialog box, 1223, 2549
Acoustic Wave Model
  Boundary, 405
acoustics model, 1205
  acoustic analogy, 1206
  broadband noise, 1206, 1231
    postprocessing, 1232
  CGNS export, 1213, 1222
  computing sound pressure data, 1219
  computing surface pressure data, 1222
  direct method, 1205
  FW-H, 1207
    postprocessing, 1219
    solving, 1219
  integral method, 1206
  quadrupoles, 1206
  reading pressure signals, 1222
  receivers, 1217
  saving source data, 1216
  source surfaces, 1214
  SYSNOISE export, 1212
  time step, 1218
  writing
    acoustic signals, 1219
    data files, 1211, 1216
Acoustics Model dialog box, 1208, 2289
Activate Cell Zones dialog box, 221, 2662
activating
  cells in parallel, 221
  zones, 221
active cell partition, 2040
active cell partition, definition, 1959
adaption, 134, 1681
  anisotropic, 1697
  boundary, 1685
  boundary layer redistribution, 1698
  curvature, 1959
  deleting registers, 1703
  display options, 1704
  displaying registers, 1704
  dynamic gradient, 1690
  eligibility, 1684
  example, 1682
  function, 1959
  geometry-based, 149, 1698
  gradient, 1688
  guidelines, 1684
  isovalue, 1691, 1959
  limiting, 1705
  manipulating registers, 1701
  modifying registers, 1703

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

region, 1692
space gradient, 1959
volume, 1694
y+ and y*, 1695
Adaption Display Options dialog box, 1704, 2663
Adaptive Time Step Settings dialog box, 2544
adaptive time stepping, 1616, 1618
adding text to the graphics window, 1789
adjacency, 224
Adjoint Observables dialog box, 2084
Adjoint Reporting dialog box, 2106
Adjoint Residual Monitors dialog box, 2100
Adjoint Solution Controls dialog box, 2092
Adjoint Solution Methods dialog box, 2091
adjoint solver
deformation region, 2108
design tool, 2107
discrete solver, 2074
discrete versus continuous solver, 2073
modifying the geometry, 2107
observable definition, 2084
overview, 2067
postprocessing, 2101
residual monitors, 2100
running the calculation, 2101
scope of functionality, 2082
shape modification, 2117
smoothing and mesh morphing, 2080
solution controls, 2092
solution methods, 2091
solution-based adaption, 2078
stabilization scheme settings, 2094
dissipation scheme, 2099
modal scheme, 2094
spatial scheme, 2097
using, 2081
using data to improve design, 2079
Advanced Solution Controls dialog box, 1566, 1576, 1578, 1581-1582, 1584, 1588, 2513
aerodynamic noise, 1205
agglomerating cells, 199
skewness-based approach, 204
aggregative multigrid solver, 1578
alphanumeric reporting, 1885
AMG
accelerating, 2056
angle of internal friction, 1459
angular coordinate, definition, 1960
angular discretization, 864
angular velocity, 320, 606, 616, 619
coordinate-system constraints, 605
Animate dialog box, 1825, 2567
animation, 1647, 1815, 1825
automatic, 1647
definition, 1647
example, 1821-1822
of pathlines, 1338, 1773
options, 1798
playback, 1826
restrictions, 1829
saving, 1828
solution, 1647
Animation Definition dialog box, 1647, 2666
Animation Recording Options dialog box, 2668
anisotropic
adaption, 1697
Anisotropic Adaption dialog box, 1697, 2671
Anisotropic Conductivity - Principal Components dialog box, 2331
Anisotropic Conductivity dialog box, 2331
anisotropic diffusion UDS, 509-510
anisotropic thermal conductivity, 500
principal axes and principal values, 506
user-defined, 507
Anisotropic UDS Diffusivity dialog box, 510
Annotate dialog box, 1790, 2592
annotated text, 1789
ANSYS CFD-Post
data file quantities for, 107
exporting data to, 69, 74, 91
during a transient calculation, 86
state files, 74, 91
ANSYS FIDAP files
importing, 64
ANSYS FIDAP neutral files, 145
ANSYS Fluent model compatibility, 2847
ANSYS Meshing
mesh files, 135
reading, 44
append case file, 145
append data file, 145
ARIES, 140
Arrhenius reaction rate, 979
ASCII files
exporting, 69, 74
during a transient calculation, 86
aspect ratio, 129, 132, 134
asynchronous, 45, 52
augmented heat transfer, 373
Aungier-Redlick-Kwong equation, 538
Auto Create Options dialog box, 164, 2672
Auto Partition Mesh dialog box, 2031, 2672
autoignition model
engine ignition, 1142

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ignition delay model, 1143-1144
knock model, 1144-1145
Autoignition Model dialog box, 1143, 2246
automatic export definitions
for particle history data, 89
for solution data, 87
Automatic Export dialog box, 87, 2530
automatic file exporting, 86
automatic file numbering, 43
automatic file saving, 48, 641, 743, 1612
automatic mesh display, 22
Automatic Particle History Data Export dialog box, 89, 2533
automatic partitioning, 2031
Automatic Solution Initialization and Case Modification dialog box, 2536
automotive exhaust systems, 373
automotive underbody simulations, 373
automotive underhood simulations, 215, 373, 858
autosave
maximum number of files, 48
Autosave Case During Mesh Motion Preview dialog box, 2502
Autosave dialog box, 48, 2527
average number of particles in soot aggregate, definition, 1960
average pressure specification
pressure outlets, 342
AVS files
exporting, 69, 74
during a transient calculation, 86
axes
attributes, 1852
XY plots, 1852
Axes dialog box, 1852, 2608
axes, principal, 506
axial coordinate, definition, 1960
axial force, 1863
axial pull velocity, definition, 1960
axial velocity, 319
axial velocity, definition, 1960
axis boundary condition, 389
Axis dialog box , 2392
axis of rotation, 228, 255, 260, 605-606, 616, 619
axisymmetric cases, 130, 228, 389, 589, 1885
mesh checking, 192
axisymmetric flow
in multiple reference frames, 614
modeling with swirl or rotation, 589
rotation axis, 605
axisymmetric mesh setup, 2162
axisymmetric swirl flows, 311, 319, 588-589
inputs for, 590
postprocessing for, 593
solution setup for, 591
solution stability of, 592
solution strategies for, 591

B

backflow, 338, 340, 353, 355
background color, 105
Backward Reaction Parameters dialog box, 2337
batch execution, 1638
Batch Options dialog box, 2659
beam irradiation flux, definition, 1960
Bernoulli equation, 315
Bernstein polynomials
adjoint solver, 2080, 2120
mesh morpher/optimizer, 2122
bi-conjugate gradient stabilized method (BCGSTAB), 1578
Biaxial Conductivity dialog box, 502, 2327
biaxial thermal conductivity, 502
binary diffusivity, 1318
binary files, 41
Bingham plastics, 495
black body emission factor, 847, 865
black body temperature, 869
Blake-Kozeny equation, 274
blending
first-to-higher order, 1555
blocking surfaces, 858
blowers, 628
body forces, 1377, 1556
for multiphase flow, 1377
implicit treatment of, 1377
boiling
point, 1318
pressure dependent, 1251
booleans , 31
boundary adaption, 1685
Boundary Adaption dialog box, 1685, 2673
boundary cell distance, 1686
boundary cell distance, definition, 1960
boundary conditions, 237
axis, 389
case check, 1661
centerline, 389
changing zones, 239
compressible flow, 597
copying, 242
coupling with GT-POWER, 451
coupling with WAVE, 456
degassing, 357

discrete ordinates (DO) radiation model
walls, 870-871, 873
discrete phase, 1301
energy sources, 297
Eulerian wall film model, 1542
exhaust fan, 302, 355
fan, 389
fixing values in cell zones, 292
flow inlets and exits, 302
heat exchanger, 396
ill-posed, 351
inlet, 297, 316
inlet vent, 302, 334
intake fan, 302, 336
listing, 1927
mass, 297
mass flow inlet, 302, 322
mass flow outlet, 330
momentum, 297
moving reference frame, 608
moving zones
fluids, 256
solids, 261
multiphase flow, 1391
multiple reference frames, 616
non-premixed combustion model, 1079
non-reflecting, 405
non-uniform inputs for, 243
NOx, 1169
outflow, 302, 350
outlet vent, 302, 353
overview, 237
periodic, 912
porous jump, 403
solar load model, 903
porous media, 261
premixed turbulent combustion, 1094
pressure far-field, 302, 347
calculation setup for, 349
inputs, 347
pressure inlet, 302, 308
pressure outlets, 302, 338
average pressure specification, 342
optional inputs for, 344
radiant, 384
radiation, 869
black body temperature, 869
emissivity, 869
inlets and exits, 869
Monte Carlo (MC) model, 876
S2S model, 870
walls, 870
radiator, 396
reading, 56
rotating flow, 590
saving, 56
setting, 241
multi-zone selection, 241
transient, 448
sliding meshes, 634
soot, 1201
source term input, 297
SOx, 1181
species, 379, 990
surface reaction, 380
swirling flow, 590
symmetry, 384
thermal, 384, 825
at walls, 370
transient, 448
turbulence, 807
input methods, 303
types of, 237
velocity inlet, 302, 316
writing, 56
Boundary Conditions task page, 238, 2389
boundary layers, 133, 589
converting to polyhedra, 199
deformation, 658, 737
disturbed, 373
redistribution, 1698
smoothing, 737
boundary meshes, writing, 56
boundary normal distance, 1686
boundary normal distance, definition, 1960
boundary profiles
file format, 436
interpolation methods
changing, 440
reading and writing files, 53
reorienting, 443
units of, 111
boundary volume distance, definition, 1960
boundary zones, 237
changing name of, 243
changing type of, 239
bounded plane, 1722
Bounding Frame dialog box, 1823, 2582
bounding frames, 1823
Bounding Orientation dialog box, 2112
Boussinesq model, 480, 833
limitations, 834
branch cuts, 214
breakage kernel, mixture multiphase model, 1452, 1461

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

breakup discrete phase models
KHRT model, 1292
SSD model, 1292
Taylor Analogy Breakup (TAB) model, 1292
WAVE model, 1292
Brownian force, 1292
Brucato et al. correlation, 1466
BSL k-omega model, model compatibility
model compatibility, 2847
bubbles, 1235
bulk species, 974-976
buoyancy
solutal, 1531
thermal, 1531
buoyancy-driven flows, 833
boundary conditions, 308
inputs for, 834
operating density, 836
postprocessing, 838
solution procedures, 837
solution strategies, 837
burnout stoichiometric ratio, 1049, 1318
burnt mixture
species concentration, 1090, 1098
button functions, 1800
modifying, 1801

C

C-equation, 1092
C-type meshes, 214
cache buffers, clearing, 2066
Calculation Activities task page, 86, 2525
calculations, 1600-1601, 1608
adjoint solver, 2101
convergence of, 1583, 1668
executing commands during, 1638
interrupting, 1601
pseudo transient, 1601
stability of, 1667, 1669
steady-state, 1600
time-dependent, 1608
Camera Parameters dialog box, 1804, 1808, 2587
canceling
a display operation, 5
a task, 18, 1600-1601
captions, 1788
carpet plot, 1759
Carreau model, 494
Carreau Model dialog box, 494, 2324
carrier, 999
Cartesian coordinate system
for boundary condition inputs, 311
Cartesian velocities, 1933
for boundary condition inputs, 318
case check, 1654
boundary conditions, 1661
grid, 1657
material properties, 1664
model, 1659
solver, 1665
Case Check dialog box, 2543
case files
automatic saving of, 48, 641, 743, 1612
Fluent 4, 68
FLUENT 4, 144
Fluent UNS, 53
format, 2851
RAMPANT, 53, 144
reading, 45-47
writing, 45-47, 206
case modification, 1642, 2536
catalytic converter, 277
cavitation model
noncondensable gases, 1455
surface tension coefficient, 1455
vaporization pressure, 1455
cell
acoustic courant number, 1961
agglomeration, 199
skewness-based approach, 204
aspect ratio, 129, 132, 134
children, 1960
convective courant number, 1961
courant number
acoustic, 1961
convective, 1961
element type, 1961
equiangular skewness, 1961
equivolume skewness, 1961
orthogonal quality, 131, 134, 1980, 2125
mesh morpher/optimizer, 2125
partition, 1962
refine level, 1962
Reynolds number, 1962
skewness, 129, 134
equiangular, 1961
skewness equivolume, 1961
surface area, 1962
types, 115
values
for contours, 1761
for pathlines, 1780
for XY plots, 1844
volume, 135, 1962

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2D, 1962
change, 135, 1962
volume derivative, 1962
volume error, 1962, 1969
wall distance, 1962
warpage, 1962
zone type, 1963
zones, 1963
activating, 221
copying, 222
deactivating, 220
deleting, 219
modifying, 218
replacing, 218
separating, 208, 210
cell acoustic courant number, 1961
cell aspect ratio, 1573
cell convective courant number, 1961
cell registers, 1736
boundary, 1738
field variable, 1739
manage, 1743
operations, 1744
region, 1737
residuals, 1740
variable limiter, 1739
volume, 1741
Yplus/Ystar, 1742
cell values
flow variables, 1931
cell zone conditions, 237
changing zones, 239
copying, 242
overview, 237
sliding meshes, 634
species, 990
types of, 237
Cell Zone Conditions task page, 2367
cell zone remeshing method, 676
cell zones
adjacent face zones, 224
changing type of, 239
requirements
dynamic meshes, 676
center of pressure, 1916
report, 1916
centering the display, 1805
centerline boundary conditions, 389
CFL condition, 1571
CFX files, 141
importing, 62
CGNS files, 69
exporting, 69, 75
during a transient calculation, 86
importing, 63
writing spectrum data, 1222
character strings , 31
characteristic length, 529
checking
case setup, 1654
meshes, 192
chemical mechanism, 1127
dimension reduction, 1128, 1136
dynamic mechanism reduction, 1131
chemical mechanism files, 68, 967, 1037
chemical reaction
equilibrium chemistry, 1025, 1031
non-equilibrium chemistry
rich flammability limit, 1035
reversible, 987
chemical vapor deposition, 995
chemistry agglomeration, 1136
chemistry model
detailed, 1202
CHEMKIN files, 68, 967, 1037, 1202
CHEMKIN Mechanism Import dialog box, 967, 995, 2642
CHEMKIN-CFD solver
diagnostic files and error messages, 2875
parameters, 2875
child droplets, 1253
chimera meshes, 172
circumferential average, 624, 639, 1848
clearing the file cache buffers, 2066
cloud tracking, 1296
clustering, 849
display, 883
S2S radiation model, 853
defining automatically, 855
defining manually, 853
Coal Calculator dialog box, 2232
coal combustion, 1032, 1049
carbon conversion, 1343
char burnout, 2354
devolatilization, 1321
discrete phase, 1268
empirical fuel definition in, 1034, 1045
fuel composition, 1046
heating value, 1045, 1047
in non-premixed combustion model, 1044, 1046
models for, 1321
water content, 1047, 1049, 1288
coalescence kernel, mixture multiphase model, 1452, 1461
coalescence, DPM model, 1252

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

coarse grid levels, 1576, 1583
coarsening
  parameters, 1580
  pathlines, 1775
coefficient of restitution, 1305
cold flow, 991
collision
  discrete element method (DEM), 1253
    friction collision law, 1253
    rolling friction collision law, 1253
    spring collision law, 1253
    spring-dashpot collision law, 1253
collision rate
  discrete phase, 1965
color
  background, 105
colormap, 1791
  alignment, 1789
  custom, 1795
  disabling, 1788
  labels, 1793
Colormap dialog box, 1791, 2589
Colormap Editor dialog box, 1795, 2591
colors, 1818, 2610
  contour, 1868-1869
  contours, 1757
  for annotation, 1790
  mesh, 1752
  pathline, 1775
  vector, 1768-1769
combustible fraction, 1049, 1319
combustion, 991
  applications, 573
  coal, 1032, 1046
  empirical fuel definition, 1034
  equilibrium chemistry, 1025, 1031
  liquid fuel, 1032
  model compatibility, 2847
  non-equilibrium chemistry
    rich flammability limit, 1035
  partially premixed, 1099
    in cylinder, 1112
  pollutant formation, 1053, 1153
  premixed, 1089
  solution techniques, 1670
  sulfur in, 1044
command files, 1638
command line history , 29
command line options
  parallel
    Linux, 2026
    Windows, 2021
command macros, 1640
  during mesh morpher/optimizer runs, 2125
commands
  abbreviations, 28
  aliases, 30
  scheme, 29
compass optimizer, 2123
Compiled UDFs dialog box, 2675
composites, thermal conduction in, 507
composition averaging coefficient, 1320
composition PDF transport model
  Eulerian
    boundary conditions, 1119
    enabling, 1118
  Lagrangian
    enabling, 1115
  model compatibility, 2847
  particle tracking, 1124
  reporting options, 1123, 1125
compressed files, 41
compressed row format (CRF), 860
  converting to, 863
compressibility effects, 755, 799
compressibility factor, definition, 1963
compressible flows, 593, 1551
  boundary conditions for, 597
  calculations at inlet pressure boundaries, 316, 1662
  equations for, 595
  floating operating pressure
    enabling of, 598
    limitations, 597
    monitoring, 598
    overview, 597
    setting initial value, 598
    theory, 597
  gas law equation, 595
  higher-order density interpolation, 1557
  inputs for, 485, 596
  model scope, 594
  physics of, 595
  pressure interpolation, 1556
  reporting results of, 599
  solution strategies, 599
Compressible Liquid dialog box, 2321
compressive scheme
  phase localized, 1443
computational expense, 129, 1681
compute cluster package (CCP), 2023
compute nodes, 2009
  connectivity information for, 2059
  latency and bandwidth information for, 2062
computing

sound pressure data, 1219
surface pressure data, 1222
view factors, 851
accelerating, 2064
limitation, 860
concentration
discrete phase, 1965
concrete, 496
condensation, 1307
conduction, 825
anisotropic, 500
principal axes and principal values, 506
user-defined, 507
shell, 376, 838
file format, 2868
initialization, 1593
limitations, 839
locking the temperature, 297
managing, 840
physical treatment, 838
postprocessing, 843
with the S2S radiation model, 851
conductive heat transfer, 825
conical mesh interface, 631
conjugate gradient method (CG), 1578
conservation equations
source term additions to, 297
conservative coarsening, 1580
consistency index, 495
console, 10
contact angle, 1393-1394, 1440
contact detection option
settings, 711
contact resistivity, definition, 1963
continuous casting
inputs for, 1530
contour
storing, 1762
contour plotting, 1756
definitions, 1762
for turbomachinery, 1867, 1869
Contours dialog box, 1757, 2676
control points, 2122, 2125
Convection/Diffusion Model dialog box, 1324, 2320
convective augmentation factor, 373
convective flux
transport equations, 574
conventions used in this guide, lxix
convergence, 837, 991, 1624, 1667
acceleration of, 1576-1577, 1584, 1591
criteria
choosing, 1629
modifying, 1631
judging, 1624, 1635, 1668
monitoring, 1635
convergence acceleration
stretched meshes, 1573
convergence conditions, 1635
Convergence Conditions dialog box, 2678
conversion factors for units, 114
converting meshes, 199
coordinates
Cartesian, 2002
constraints for moving reference frames, 605
radial, 1984
Copy Case Material dialog box, 2315
Copy Collision Partner dialog box , 2286
Copy Conditions dialog box, 242, 2381
Copy From dialog box, 2206
copying
zones, 222
core porosity model
default, 941
defining, 941
effectiveness, 941
reading parameters from an external file, 942
Core Porosity Model dialog box, 941, 2211
cores, parallel processing, 2058
Cortex, 30
coupled flow and heat transfer, 829
coupled level set, 1409
coupled pressure-based solver, 1563
Eulerian multiphase model, 1507
mixture multiphase model, 1507
VOF multiphase model, 1507
coupled walls, 215, 375
interface option, 156, 164
Courant number, 1577
density-based explicit formulation, 1571
density-based implicit formulation, 1571
for VOF calculations, 1445
setting, 1571
Coverage-Dependent Reaction dialog box, 2341
CPD Model dialog box, 2352
crank angle
appending to file name, 48
Create Collision Partner dialog box , 2286
Create New Condition dialog box, 2112
Create New Observable dialog box, 2085
Create Surface dialog box, 1782, 2566
Create/Edit Materials dialog box, 972, 1080, 2302
Create/Edit Mesh Interfaces dialog box, 170, 2680
Create/Edit Overset Interfaces dialog box, 2470
creating

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

meshes, 135
crevice model
engine ignition, 1145
output file, 1150
postprocessing, 1148
CRF, 860
critical pressure, definition, 1963
critical specific volume, definition, 1963
critical strain rate, definition, 1963
critical temperature, definition, 1963
Cross model, 495
Cross Model dialog box, 495, 2325
crossflow instability, 800
cubic equation of state real gas model, 535, 542
Aungier-Redlich-Kwong equation, 536, 538
Lagrangian dispersed phase model, 548
limitations, 535
Peng-Robinson equation, 536, 538
Postprocessing, 550
Redlich-Kwong equation, 535, 537
Soave-Redlich-Kwong equation, 536, 538
current density, definition, 1963
curvature correction, 798
curvature correction function fr, definition, 1963
curve fitting, 1194
Curves dialog box, 1854, 2610
Custom Field Function Calculator dialog box, 2004, 2683
custom field functions, 2003
definition, 1964
exporting, 2003
for solution initialization, 1595
for unsteady statistics, 1613
postprocessing, 1623
sample, 2007
saving, 2006
units of, 111, 2003
Custom Laws dialog box, 2684
Custom Vectors dialog box, 1769, 2558
Customer Portal, 26
customizing
field functions, see also custom field functions, 2003
GUI, 22
units, 112
vectors, 1769
CutCell Boundary Zones Info dialog box, 734, 2498
CutCell meshes
removing hanging nodes / edges, 143
removing hanging nodes / edges from, 205
CutCell zone remeshing method, 681
inflation layers, 683
CVD, 995
cyclic boundary, 386
cylindrical coordinate system for boundary condition inputs, 311
Cylindrical Orthotropic Conductivity dialog box, 504, 2328
cylindrical orthotropic diffusion UDS, 512
cylindrical orthotropic thermal conductivity, 504
Cylindrical Orthotropic UDS Diffusivity dialog box, 512
cylindrical velocities, 1933
for boundary condition inputs, 318

D

Damkohler number, definition, 1964
Darcy's Law, 262-263, 403
data
exporting, 69
after a calculation, 70
during a transient calculation, 86
limitations, 69
steady-state, 84
importing, 59, 1338
mapping
fluid-structure interaction (FSI) problems, 97
resetting, 1601
Data Explorer files
exporting, 69, 76
during a transient calculation, 86
Data File Quantities dialog box, 107, 2529
data files
automatic saving of, 48, 641, 743, 1612
Fluent UNS, 53
format, 2851
options, 107
parallel, 51
particle
importing, 1338
particle history
exporting, 84, 86
postprocessing
time-sequence data, 1873
with alternative applications, 107
quantities saved in, 107
RAMPANT, 53
reading, 45, 47
writing, 45, 47, 206
Data Sampling for Time Statistics
enabling, 1613
database
materials, 460, 1316
data sources, 464
user-defined, 467
DCC, 1137

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DDES
see also delayed detached eddy simulation (DDES), 800
Deactivate Cell Zones dialog box, 220, 2685
deactivating
cells, 220
zones, 220
parallel, 220
Decoupled Detailed Chemistry dialog box, 1202, 2273
decoupled detailed chemistry model, 1202
decoupled flow and heat transfer, 829
Define
Materials..., 508
Define Control Points dialog box, 2125, 2685
Define Event dialog box, 713, 2487
Define Macro dialog box, 1640, 2536
Define Unit dialog box, 113, 2167
define/boundary-conditions/ , 241
define/boundary-conditions/modify-zones/create-all-shell-threads , 376
define/boundary-conditions/modify-zones/delete-all-shells , 376, 840
define/boundary-conditions/non-reflecting-bc , 414
define/dynamic-mesh/actions/remesh-cell-zone , 676
define/dynamic-mesh/actions/remesh-cell-zone-cutcell , 684
define/dynamic-mesh/controls/remeshing-parameter/remeshing-after-moving? , 670
define/dynamic-mesh/controls/remeshing-parameters/zone-remeshing , 668
define/dynamic-mesh/controls/smoothing-parameters/smooth-from-reference-position?, 657
define/mesh-morpher-optimizer/optimizer-parameters/disable-mesh-check , 2125
define/mesh-morpher-optimizer/region/scaling-enlarge , 2125
define/mesh-morpher-optimizer/region/scaling-reduce , 2125
define/mixing-planes/set/conserve-swirl , 623
define/mixing-planes/set/fix-pressure-level , 622
define/models/acoustics/auto-prune , 1221
define/models/acoustics/cylindrical-export? , 1212
define/models/acoustics/export-volumetric-sources? , 1212
define/models/acoustics/write-centroid-info , 1212
define/models/dpm/options/track-in-absolute-frame , 614, 1238
define/models/heat-exchanger/macro-model/heat-exchanger-macro-report , 955
define/models/multiphase/wet-steam/compile-user-defined-wetsteam-functions , 1495, 1498
define/models/multiphase/wet-steam/load-unload-user-defined-wetsteam-library , 1495
define/models/multiphase/wet-steam/set/max-liquid-mass-fraction , 1518
define/models/radiation/do-acceleration? , 2065
define/models/radiation/s2s-parameters/compute-clusters-and-vf-accelerated , 2064
define/models/radiation/s2s-parameters/split-angle , 856
define/models/radiation/solar-parameters/autoread-solar-data/ , 907
define/models/radiation/solar-parameters/autosave-solar-data/ , 907
define/models/shell-conduction/enhanced-encapsulation? , 2056
define/models/species/ecfm-controls , 1095
define/models/species/inlet-diffusion? , 1079
define/models/viscous/near-wall-treatment/werner-wengle-wall-fn? , 792
define/models/viscous/turbulence-expert/turb-non-newtonian? , 804
define/parameters/input-parameters/advance/use-in , 249
define/user-defined/compiled-functions , 567
DEFINE_SOURCE_AE, 2083
definitions
automatic export
for particle history data, 89
for solution data, 87
for flow variables, 1957
Deform Adjacent Boundary Layer with Zone , 658
deformation regions, 2122, 2125
deforming zones, 722, 731
inflation layers, 734
degassing boundary condition, 357
Degassing dialog box , 2392
degrees of freedom, 529
delayed detached eddy simulation (DDES), 800
Delete Cell Zones dialog box, 219, 2687
deleting
objects from a scene, 1823
zones, 219
Delta-Eddington Scattering Function dialog box, 2349
Delta-Eddington scattering phase function, 518
DEM Collision Settings dialog box , 2287
DEM Collisions dialog box , 2285
DEM model
see also discrete element method, 1253
Dennis-et-al, 1291
dense discrete phase model
postprocessing, 1523
density, 479, 986

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Boussinesq model, 479
composition-dependent, 486
constant, 479-480
definition, 1964
discrete phase, 1321
ideal gas law, 479, 485
incompressible ideal gas law, 479, 484
interpolation schemes, 1557
restrictions, 480
temperature-dependent, 479, 484
density-based solver, 1551
Courant number, 1571
explicit
Courant number, 1571
explicit relaxation, 1575
flux types, 1573
implicit
Courant number, 1571
implicit formulation
convergence acceleration, 1573
limitations, 1551
residuals, 1626
derivatives, definition, 1964
DES length scale, definition, 1964
DES TKE dissipation multiplier, definition, 1964
DES turbulence model
see also detached eddy simulation (DES) turbulence model, 758
design analysis and optimization, 2067
Design Tool dialog box, 2107
detached eddy simulation (DES) turbulence model, 758
curvature correction, 798
intermittency transition model, 800
detailed chemistry model
decoupled, 1202
devolatilization, 1966
models, 1321
diesel engines
ignition delay model, 1143-1144
diffusion, 129, 508
species, 829
thermal, 1996
diffusion coefficient, 959, 986
multiphase flow, 574
single phase flow, 574
species
effective, 1969
laminar, 1976
thermal, 522
user-defined scalar, 1964
diffusion-based smoothing method, 650
applicability, 655
diffusivity based on boundary distance, 653
diffusivity based on cell volume, 655
setting up, 644
diffusive heat flux, 373
diffusivity
binary, 1318
diffusivity reference pressure, 1322
dimension reduction, 1128, 1136
direct search methods, 2123
directed relation graph (DRG) method, 1131
disabling graphics windows, 1783
discrete element method (DEM)
collision, 1253
friction collision law, 1253
rolling friction collision law, 1253
spring collision law, 1253
spring-dashpot collision law, 1253
discrete ordinates (DO) radiation model
accelerating, 2065
angular discretization, 864
boundary conditions
opaque walls, 871
semi-transparent walls, 873
walls, 870-871
energy coupling
enabling, 866
GPGPUs, 2065
material properties, 868
non-gray
absorption coefficient, 869
black body emission factor, 865
emissivity weighting factor, 865
inputs for, 865
refractive index, 869
parallel processing, 2065
properties, 515
residuals, 881
setting up, 864
solution parameters, 880
discrete phase, 1235
aborted trajectories, 1246
absorption coefficient, 1965
accretion, 1251, 1309, 1965
reporting, 1360
air-blast atomizer
injections, 1278
boundary conditions, 1301
escape, 1303
interior, 1303
reflect, 1301
wall film, 1303
wall jet, 1303

breakup model, 1253
Brownian force, 1292
burnout, 1965
cloud tracking, 1296
coal combustion, 1049
coefficient of restitution, 1301-1302, 1305
collision, 1252
  discrete element method (DEM), 1253
combusting, 1268, 1316
concentration, 1965
condensate
  injections, 1281
condensation, 1307
cone injections, 1266, 1271
coupled calculations, 1327, 1329
coupled heat-mass solution, 1262
customized particle laws, 1288, 1296
devolatilization, 1966
discrete element method (DEM)
  collision model, 1253
display, 1330
drag coefficient, 1290
droplet, 1268, 1316
droplet coalescence, 1252
droplet temperature
  latent heat, 1252
effervescent atomizer
  injections, 1280
emission, 1965
erosion, 1251, 1309, 1966
  Finnie, 1966
  McLaury, 1966
  reporting, 1360
evaporation, 1966
file
  injections, 1280
flat-fan atomizer
  injections, 1278
friction coefficient, 1305
group injections, 1266, 1270
heat transfer, 1240, 1343
in multiple reference frames, 614, 1238
inert, 1268, 1316
initial conditions, 1265
  file, 1266
  read from a file, 1268
injections, 1266
  copying, 1286
  creating, 1285
  deleting, 1286
  file, 1266
  listing, 1286
  read from a file, 1268
  reading, 1286
  setting properties, 1286
  setting properties for multiple, 1297
  writing, 1286
inputs, 1240
  for boundary conditions, 1301
  for initial conditions, 1265
  for material properties, 1313
  for optional models, 1249
  for particle tracking, 1241
  for transient particles, 1242
interaction with continuous phase, 1241
interphase exchange, 1351
lift force, 1251
limitations of, 614, 1238
linearize source terms, 1264
mass flow rate, 1269, 1273, 1283
mass transfer, 1240, 1318, 1343
materials, 461, 1288, 1313, 1316
model compatibility, 2847
multicomponent, 1268, 1316
node based averaging, 1263
numerics, 1260
options, 1249
overview, 1240
parallel processing, 1360
  hybrid, 1360
  message passing, 1360
  shared memory, 1360
  workpile, 1360
parameter tracking, 1246
parcels, 1242
particle cloud tracking, 1296
particle rotation, 1352
  Dennis-et-al, 1291
  none, 1291
particle stream, 1286
particle summary reporting, 1358
particle tracking, 1240
plain-orifice atomizer
  injections, 1276
point properties, 1269
postprocessing, 1329
pressure dependent boiling, 1251
pressure gradient force, 1251
pressure-swirl atomizer
  injections, 1277
properties, 1313
radiation heat transfer to, 1250, 1324
reference frame, 1263
residence time, 1343

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

restitution coefficient, 1301-1302, 1305
rough wall model, 1292
scattering, 1968
secondary breakup models
KHRT model, 1292
stochastic secondary droplet (SSD) model, 1292
Taylor Analogy Breakup (TAB) model, 1292
WAVE model, 1292
setup, 1284
size distribution, 1281
solution procedures, 1325
sources, 1351, 1966
staggering, 1264
stochastic tracking, 1294, 1328
summary report, 1341
surface injections, 1266, 1274
thermophoretic coefficient, 1323
thermophoretic force, 1251
time step, 1246
tracking schemes, 1261
trajectory calculations, 1325
trajectory reporting, 1340, 1347
at boundaries, 1355
unsteady tracking, 1349
transient particles, 1242
turbulence, 1252
uncoupled calculations, 1326
under-relaxation, 1328
unsteady statistics, 1353
user-defined functions, 1258, 1296
using, 1240
variables, 1351
virtual mass force, 1251
wall boundary layer model, 1309
wet combustion, 1288
Discrete Phase dialog box, 1484, 2361
Discrete Phase Model dialog box, 1246, 1249, 1284, 1331, 2278
discrete phase sources, definition, 1964
discrete phase variables, definition, 1965
discrete random walk (DRW) model, 1294
discrete solver, adjoint, 2074
discrete transfer radiation model (DTRM)
clusters
controlling, 849
displaying, 851
properties, 515
rays
controlling, 850
defining, 848
reading and writing files, 850
residuals, 881
setting up, 848
solution parameters, 879
discretization, 129
first-order scheme, 1555
first-to-higher order blending, 1555
inputs for, 1554
power-law scheme, 1556
QUICK scheme, 1556
second-order scheme, 1555
Display Options dialog box, 1786, 1788, 1796, 1798, 2573
display properties, 1817
Display Properties dialog box, 1817, 2578
display/set/title , 1788
display/set/windows/scale , 1788
display/set/windows/text , 1788
display/set/zero-angle-dir , 1856
displaying
animations, 1825
restrictions, 1829
annotations, 1789
bounding frames, 1823
captions, 1788
colormap, 1791
colors, 1818
contours, 1756
for turbomachinery, 1867, 1869
creating surfaces for, 1709
elements
poor, 1977
graphics, 1747
in multiple graphics windows, 1786
legends, 1788
lights, 1796
meshes, 1748
automatically, 22
mirrored domains, 1811
on surfaces with uniform distribution, 1718, 1722
on sweep surfaces, 1782
outlines, 1748, 1753
overlaid graphics, 1785
pathlines, 1772
periodic repeats, 1811
poor elements, 194
profiles, 1756
properties, 1817
rendering options, 1798
scenes, 1815
symmetry, 1811
titles, 1788, 1845
transparency, 1818
turbomachinery

2D contours, 1869
averaged contours, 1867
vectors, 1764
views, 1803
visibility, 1818
dissipation, 1589
disturbed boundary layers, 373
divergence, 1564
troubleshooting, 1587
domain, reordering, 226
donor-acceptor scheme, 1373
double buffering, 1798
double-precision solvers, 1624
dough, 496
DPM Collision Rate, definition, 1965
DPM conc of <component>, definition, 1965
DPM D20, definition, 1967
DPM D30, definition, 1967
DPM D32, definition, 1967
DPM density, definition, 1965
DPM diameter, definition, 1965
DPM parcels in cell, definition, 1967
DPM particles in cell, definition, 1967
DPM Report Definition dialog box, 2687
DPM RMS diameter, definition, 1967
DPM RMS temperature, definition, 1967
DPM RMS X, Y, Z velocity, definition, 1967
DPM Source Report Definition dialog box, 2690
DPM specific heat, definition, 1968
DPM temperature, definition, 1968
DPM volume fraction, definition, 1968
DPM X, Y, Z velocity, definition, 1969
drag coefficient, 1916
discrete phase, 1290
drag factor, 1466
drag modification, 1466
drag report definition, 1893
Drag Report Definition dialog box, 2691
DRG reduced number of reactions, definition, 1969
DRG reduced number of species, definition, 1969
droplet, 1235, 1268, 1316
breakup, 1253
coalescence, 1252
collision, 1252
discrete element method (DEM), 1253
initial conditions, 1265
size distribution, 1281
trajectories
alphanumeric reporting, 1351
display of, 1330
droplet temperature
latent heat, 1252
DRW model, 1294
DTRM Graphics dialog box, 883, 2694
DTRM Rays dialog box, 848, 2695
Dual Cell Heat Exchanger dialog box, 922, 2201
dual cell heat exchanger model, 920-921
restrictions, 922
duals, 199
duplicate shadow nodes, 194
dynamic cell clustering, 1137
dynamic head, 396
dynamic layering method, 661
dynamic mechanism reduction, 1131
Dynamic Mesh Events dialog box, 713, 2486
dynamic mesh model
model compatibility, 2847
partitioning in parallel, 2039, 2043
Dynamic Mesh task page, 643, 2471
Dynamic Mesh Zones dialog box, 723, 2490
dynamic meshes
2.5D model, 686
boundary layer smoothing, 658, 737
constraints, 649, 664, 669, 676, 680, 683, 686, 690
contact detection
settings, 711
crevice model, 1145
defining events, 713
diffusion-based smoothing method, 650
applicability, 655
diffusivity based on boundary distance, 653
diffusivity based on cell volume, 655
setting up, 644
dynamic layering, 661
dynamic layering method, 661
implicit update, 643
settings, 709
in-cylinder motion, 695
in-cylinder option, 643
piston pin offset, 690
settings, 690
inflation layers, 681
Laplacian smoothing, 644, 657
linearly elastic solid based smoothing method, 656
applicability, 656
mesh motion methods, 644
feature detection, 690
volume mesh update, 689
mesh requirements
2.5D surface remeshing, 686
cell zone remeshing, 676
CutCell zone remeshing, 683
dynamic layering, 664
face region remeshing, 680

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

feature detection, 690
local face remeshing, 669
spring-based smoothing, 649
periodic motion, 657
previewing, 742
remeshing, 665
remeshing methods, 665
2.5D surface, 685
cell zone, 676
CutCell zone, 681
face region, 676, 678
local, 668
local cell, 669
local face, 669
setting parameters, 643
setup, 641
six DOF solver, 643, 705
one DOF motion, 707
sets of properties, 707
settings, 705
smoothing, 644
from a reference position, 657
periodic motion, 657
solid-body kinematics, 739
specifying zone motion, 722
spring-based smoothing, 644
spring-based smoothing method, 646, 649
setting up, 644
steady-state, 744
additional local remeshing, 670
dynamic pressure, 1969

E

ECFM spark model, 1141
EDC model, model compatibility, 2847
eddy-dissipation model, model compatibility, 2847
Edit Automatic Initialization and Case Modifications dialog box, 1647
Edit Material dialog box, 2355
Edit Mesh Interfaces dialog box, 164, 2696
Edit Property Methods dialog box, 2316
Edit Report File, 1905
Edit Report File dialog box, 2698
Edit Report Plot, 1908
Edit Report Plot dialog box, 2700
effective Prandtl number, 1969
effective thermal conductivity, definition, 1969
effective viscosity, definition, 1969
effectiveness
core porosity heat exchanger model, 941
efficiencies
for pumps and compressors, 1864
for turbines, 1866
EGR, 1112
inert model
resetting, 722, 1077
elasticity modulus, 1450, 1460, 2360, 2363
electric conductivity, definition, 1969
electric potential model
limitations, 1547
options, 1547
overview, 1547
postprocessing variables, 1549
electric potential, definition, 1969
electrochemical reactions, 1003
inputs, 1004
solution, 1011
sources
continuity equation, 1011
energy equation, 1010
transport equation, 1011
electrode surface potential, definition, 1970
elements
poor, 194, 1977
Embedded LES (ELES) turbulence model, 760
emissivity, 869-870
emissivity weighting factor, 847, 865
empirical fuel, 1034, 1045
encapsulation, 2056
encrypted mechanism, 967
energy, 1975, 1997
parameter, 529
sources, 299
Energy dialog box, 596, 825, 834, 2175
energy source terms
defining, 299
engine ignition
autoignition model, 1142
crevice model, 1145
spark model, 1139
EnSight Case Gold files
exporting, 69, 76
during a transient calculation, 86
EnSight files
importing, 64
pathlines, 1779
EnSight particle files
exporting
during a transient calculation, 86
from a steady-state solution, 84
enthalpy
definition, 1970
sensible, 1988
standard-state, 987

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

entropy, 987, 1924
  definition, 1971
equation of state, 537
Equations dialog box, 1669, 2511
equilibrium chemistry, 1025, 1031
  condensed phase species, 1045
  empirical fuels, 1035
Ergun equation, 274
erosion, 1251, 1309, 1966
  Finnie, 1310, 1966
  generic erosion model, 1309
  McLaury, 1310, 1966
  Oka, 1311
  reporting, 1360
error reduction, 1555
Eulerian multiphase model
  drag function, 1463
    modification, 1466
  lift forces, 1466
  model compatibility, 2847
  restitution coefficients, 1466
  solution method
    coupled pressure-based solver, 1507
  solution strategies, 1455, 1516
  turbulent dispersion force, 1470
  virtual mass force, 1474
  wall lubrication force, 1467
Eulerian Wall Film dialog box, 2294
Eulerian wall film model
  boundary conditions, 1542
  initial conditions, 1542
  limitations, 1535
  options, 1535
  postprocessing variables, 1545
  reporting mass flow rate, 1545
  solution controls, 1539
evaporation, 1966
events
  defining dynamic mesh, 713
Events Preview dialog box, 713, 2489
Execute Commands dialog box, 1638, 2535
Execute on Demand dialog box, 2702
exhaust fan boundary, 302, 355
Exhaust Fan dialog box, 355, 2393
exhaust systems, 373
existing value, definition, 1971
explicit relaxation
  density-based solver, 1575
explicit scheme, 1373
explicit time stepping
  restrictions, 1575
exploded views, 1820
Export dialog box, 70, 2644
Export Particle History Data dialog box, 84, 2648
Export STL dialog box, 2117
Export to CFD-Post dialog box, 2650
exporting
  ABAQUS files, 69, 71, 86
  ANSYS CFD-Post compatible files, 69, 74, 91
    during a transient calculation, 86
  ASCII files, 69, 74, 86
  AVS files, 69, 74, 86
  CGNS files, 69, 75, 86
  Data Explorer files, 69, 76, 86
  EnSight Case Gold file, 69, 76, 86
  EnSight particle files
    during a calculation, 86
    from a steady-state solution, 84
  FAST files, 69, 79, 86
  FAST Solution files, 69, 80, 86
  Fieldview Unstructured files, 69, 80, 86
  files, 69
    after a calculation, 70
    during a transient calculation, 86
    limitations, 69
  force data, 69
  heat flux data, 832
  I-deas Universal files, 69, 81, 86
  Mechanical APDL files, 69, 72
  Mechanical APDL Input files, 69, 73, 86
  NASTRAN files, 69, 82, 86
  particle history data
    steady-state, 84
    transient, 86
  PATRAN files, 69, 82, 86
  polyhedral meshes, 69
  RadTherm files, 69, 83, 86
  Tecplot files, 69, 83, 86
Expression Report Definition dialog box, 2703
extended coherent flame model, 1094, 1141
external flows, 347
  boundary conditions, 308
external heat transfer boundary conditions, 384
extruding face zones, 217

F

face area magnitude, definition, 1971
face area, definition, 2003
face handedness, 192
  definition, 1971
  repairing, 194
face node order, 192
  repairing, 194
face region remeshing method, 676

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

with prism layers, 678
face swapping, 235
face zones
adjacent to cell zones, 224
extruding, 217
fusing, 212
orienting, 217
separating, 208
slitting, 215
face-node connectivity, 121
facet values
flow variables, 1932
false diffusion, 129
Fan dialog box, 391, 2397
fan model
2D, 217, 389
postprocessing for, 1761, 1780, 1844
user-defined, 389, 427
3D, 287
file format, 2869
inputs for, 289
theory, 287
far-field boundary, 302, 320, 347
faradaic current density, definition, 1971
faradaic heat source, definition, 1971
FAST files
exporting, 69, 79
during a transient calculation, 86
Fast Fourier Transform (FFT)
acoustic sources, 1222
band analysis, 1222
customizing the input, 1877
customizing the output, 1879
defining the spectrum smoothing, 1877
export of surface pressure spectra, 1222
limitations, 1873
loading data, 1876
postprocessing, 1873
using, 1875
windowing, 1873
FAST Solution files
exporting, 69, 80
during a transient calculation, 86
fe2ram , 142
feature detection, 690
feature outline, 1753
Fick's law, 519
Field Function Definitions dialog box, 2006, 2704
field functions
custom, 2003
sample, 2007
saving, 2006
units of, 111, 2003
definitions, 1957
for solution initialization, 1595
Fieldview files, 69
pathlines saved in, 1778
Fieldview Unstructured files
exporting, 69, 80
during a transient calculation, 86
file cache buffers, clearing, 2066
file formats, 2851
3D fan curve, 2869
case and data files, 2851
exporting, 69, 86
importing, 59
mesh morpher/optimizer, 2867
shell conduction settings, 2868
File ribbon tab, 2633
Batch Options..., 2659
Data File Quantities..., 107, 2658
Exit, 2659
Export to CFD-Post..., 91, 2650
Export/During Calculation/Particle History Data..., 2650
Export/During Calculation/Solution Data..., 2650
Export/Particle History Data..., 84, 2648
Export/Solution Data..., 70, 2644
FSI Mapping/Surface..., 97, 2656
FSI Mapping/Volume..., 97, 2653
Import/ABAQUS/Filbin File..., 62, 2639
Import/ABAQUS/Input File..., 62, 2639
Import/ABAQUS/ODB File..., 62, 2639
Import/CFX, 141
Import/CFX/Definition File..., 62, 2639
Import/CFX/Result File..., 62, 2639
Import/CGNS/Data..., 63, 2640
Import/CGNS/Mesh & Data..., 63, 2640
Import/CGNS/Mesh..., 63, 2639
Import/CHEMKIN Mechanism..., 68, 967, 995, 2642
Import/EnSight..., 64, 2640
Import/FIDAP..., 64, 145, 2640
Import/Fluent 4 Case File..., 68, 2642
Import/FLUENT 4 Case File..., 144
Import/GAMBIT..., 64, 2640
Import/HYPERMESH ASCII..., 65, 2640
Import/I-deas Universal..., 65, 137, 2640
Import/LSTC/Input File..., 65, 2640
Import/LSTC/State File..., 65, 2640
Import/Marc POST..., 65, 2640
Import/Mechanical APDL/Input File..., 66, 2641
Import/Mechanical APDL/Result File..., 66, 2641
Import/NASTRAN, 138
Import/NASTRAN/Bulkdata File..., 66, 2641

Import/NASTRAN/Op2 File..., 66, 2641
Import/Partition/Metis Zone..., 2052, 2642
Import/Partition/Metis..., 2052, 2642
Import/PATRAN Neutral..., 67, 139, 2641
Import/PLOT3D Grid..., 67
Import/PLOT3D/Grid File..., 2641
Import/PLOT3D/Result File..., 2641
Import/PreBFC File..., 68, 136, 2642
Import/PTC Mechanica Design..., 67, 2641
Import/Tecplot..., 67, 2641
Interpolate..., 94, 2652
Read/Case & Data..., 47, 2636
Read/Case..., 46, 2635
Read/Data..., 47, 52, 2636
Read/DTRM Rays..., 2636
Read/ISAT Table..., 2636
Read/Journal..., 59, 2637
Read/Mesh..., 44, 2634
Read/PDF..., 2636
Read/Profile..., 54, 440, 2636
Read/Scheme..., 57, 2637
Read/View Factors..., 2636
Save Picture..., 102, 2658
Solution Files..., 2651
Write/Autosave..., 641, 2638
Write/Boundary Mesh..., 56, 2638
Write/Case & Data..., 47, 2637
Write/Case..., 46, 2637
Write/Data..., 47, 52, 2637
Write/Flamelet..., 1060, 2638
Write/ISAT Table..., 2638
Write/PDF..., 1068, 2638
Write/Profile..., 54, 2638
Write/Start Journal..., 59, 2638
Write/Start Transcript..., 59, 2639
Write/Stop Journal, 59, 2639
Write/Stop Transcript, 59, 2639
Write/Surface Clusters..., 862, 2638
File XY Plot dialog box, 1844, 2599
file/confirm-overwrite? , 44
file/export/custom-heat-flux , 832
file/export/fieldview-unstruct-data , 80
file/export/mechanical-apdl , 72
file/hdf-files?, 47
file/read-macros , 1641
file/read-settings , 56
file/read-transient-table , 451, 1300
File/Read/DTRM Rays... ribbon tab item, 850
File/Read/PDF... ribbon tab item, 1082
File/Read/View Factors... ribbon tab item, 863
file/single-precision-coordinates? , 46
file/transient-export/settings/cfd-post-compatible , 87
file/write-macros , 1641
file/write-settings , 56
file/write-surface-clusters/split-angle , 856
File/Write/Surface Clusters... ribbon tab item, 853
filenames , 31
files, 39
ABAQUS
exporting, 69, 71, 86
importing, 62
mapping data with, 97
ANSYS CFD-Post compatible, 74, 91, 107
exporting, 69
transient, 86
ANSYS FIDAP
importing, 64
ANSYS Meshing meshes, 135
reading, 44
ASCII
exporting, 69, 74, 86
asynchronous, 45, 52
automatic exporting of, 86
automatic saving of, 48, 641, 743, 1612
AVS
exporting, 69, 74, 86
binary, 41, 45
boundary mesh, 56
case, 45-46
autosaving, 48
CFX
importing, 62
CGNS
exporting, 69, 75, 86
importing, 63
writing spectrum data, 1222
chemical mechanism, 68, 967, 1037
compressed
reading, 41
writing, 42
data, 45, 47
autosaving, 48
options, 107
parallel, 51
postprocessing with alternative applications, 107
quantities saved in, 107
Data Explorer
exporting, 69, 76, 86
DTRM rays, 850
encrypted mechanism, 967
EnSight, 1779
importing, 64
EnSight Case Gold
exporting, 69, 76, 86

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EnSight particle
exporting during a calculation, 86
exporting from a steady-state solution, 84
exporting, 69
after a calculation, 70
during a transient calculation, 86
for steady-state calculations, 84
limitations, 69
polyhedral meshes, 69
FAST
exporting, 69, 79, 86
FAST Solution
exporting, 69, 80, 86
Fieldview
pathlines saved in, 1778
Fieldview Unstructured
exporting, 69, 86
FieldView Unstructured
exporting, 80
flamelet
steady premixed, 1109
Fluent Meshing meshes, 136
reading, 44
Fluent meshing mode meshes, 136
reading, 44
fluid-structure interaction (FSI), 97
format, 2851
3D fan curve, 2869
case and data, 2851
detecting, 41
mesh morpher/optimizer, 2867
shell conduction settings, 2868
formatted, 41, 45
GAMBIT
importing, 64
meshes, 136
pathlines, 1778
reading, 44
GeoMesh
importing, 64
meshes, 136
reading, 44
hierarchical data format (HDF), 47
overset cases, 191
HYPERMESH ASCII
importing, 65
I-deas Universal
exporting, 69, 81, 86
importing, 65
mapping data with, 97
importing, 59
particle data, 1338
interpolation, 96
journal, 57
log, 57
LSTC
importing, 65
Marc POST
importing, 65
Mechanical APDL
exporting, 69, 72
importing, 66
mapping data with, 97
Mechanical APDL Input
exporting, 69, 73, 86
mesh, 44
MPEG, 1653, 1829
naming
conventions, 40
options, 43
NASTRAN
exporting, 69, 82, 86
importing, 66
mapping data with, 97
numbering, 43
overwriting, 44
parallel data, 51
pathline, 1777
EnSight format, 1779
Fieldview format, 1778
GAMBIT format, 1778
PATRAN
exporting, 69, 82, 86
importing, 67
mapping data with, 97
picture, 102
PLOT3D
importing, 67
PreBFC meshes, 136
reading, 44
profile, 53
PTC Mechanica Design
importing, 67
study, 67
RadTherm
exporting, 69, 83, 86
ray, 850
ray tracing, 848
reading, 39
compressed, 41
shortcuts, 39
toolbar buttons, 44
reading multiple, 22
recently read, 41

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

searching for, 21
shortcuts for reading and writing, 39
state, 74, 91
suffixes, 40
surface mechanism, 995
surface mesh, 56
Tecplot
exporting, 69, 83, 86
importing, 67
text, 41, 45
transcript, 59
unformatted, 41, 45
writing, 39
compressed, 41
shortcuts, 39
toolbar buttons, 44
XML, 2025
filled
contours, 1759
meshes, 1752, 1818
profiles, 1759
film
Courant number, 1972
DPM energy source, 1972
DPM mass source, 1972
DPM x-momentum source, 1972
DPM y-momentum source, 1973
DPM z-momentum source, 1973
effective pressure, 1972
mass, 1971
passive scalar, 1972
shed mass, 1973
stripped diameter, 1972
stripped mass source, 1972
surface temperature, 1972
surface velocity magnitude, 1972
surface x-velocity, 1972
surface y-velocity, 1972
surface z-velocity, 1972
temperature, 1971
thickness, 1971
velocity magnitude, 1972
Weber number, 1972
x-momentum source, 1973
x-velocity, 1971
y-momentum source, 1973
y-velocity, 1972
z-velocity, 1972
filter papers, 261, 275
filters
fe2ram , 142
fl42seg , 136, 144
partition, 2052
tmerge , 148
tpoly , 143
fine scale
mass fraction, 1973
temperature, 1973
transfer rate, 1973
finite-rate reactions
electrochemical, 1003
particle surface, 1001
reacting channel model, 1012
volumetric, 958
wall surface, 995
Finnie Model Parameters dialog box, 1310
first-order accuracy, 1555
first-to-higher order blending, 1555
fixing variable values in cell zones, 292
fl42seg , 136, 144
flame speed model
Peters, 1093
Zimont, 1092
flamelet
diesel unsteady
resetting, 722
Flamelet 2D Curves dialog box, 2239
Flamelet 3D Surfaces dialog box, 1060, 2237
Flamelet Fluid Zones dialog box, 2240
flamelet model
diesel unsteady
resetting, 1041
diffusion
setting up, 1036
files
standard, 1073, 1109
flamelet generation approach
inputs for, 1036
importing a flamelet file, 1037, 1102
inputs for, 1025
look-up tables, 1068
look-up tables with Flamelet Generated Manifolds, 1107
parameters, 1054
PDF table parameters, 1063, 1105
standard format files, 1073, 1109
steady diffusion
automated grid refinement, 1057
setting up, 1036
steady premixed, 1103
files, 1109
unsteady diffusion
setting up, 1036
using, 1037

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

zeroing species, 1055
unsteady laminar
diesel, 1038
flexible-cycle multigrid, 1577
floating operating pressure, 597
Flow Controls dialog box, 2485
flow direction
at mass flow inlets, 327
at mass flow outlets, 332
at pressure far-field boundaries, 348
at pressure inlets, 311
flow distributors, 261
flow rate
for multiphase calculations, 1524
flow time
appending to file name, 48, 87
flow variable definitions, 1957
flow variables
cell values, 1931
facet values, 1932
node values, 1931
flow, perturbed, 373
Fluent 4 case files, 68
FLUENT 4 case files, 144
Fluent Database Materials dialog box, 464, 975, 2312
Fluent Launcher
parallel options, 2012
remote options, 2017
scheduler options, 2015
Fluent Meshing mesh files, 136
reading, 44
Fluent meshing mode, 1
mesh files, 136
reading, 44
reading multiple mesh files, 147
Fluent solution mode, 1
reading multiple mesh files, 146
Fluent UNS case and data files, 53
Fluent/UNS case files, 144
Fluid dialog box, 253, 267, 289, 576, 2369
fluid flow
compressible, 593
inviscid, 599
periodic, 582
swirling and rotating, 587
fluid materials, 254, 461, 975
fluid zone, 252
fluid-structure interaction (FSI) simulations, 97, 709
flux
diffusive heat, 373
reports, 1911
through boundaries, 1911
types
AUSM, 1573
low diffusion Roe-FDS, 1573
Roe-FDS, 1573
flux report, 1885
Flux Report Definition dialog box, 2705
Flux Reports dialog box, 1911, 2613
FMG multigrid, 1595
convergence strategies, 1597
using, 1596
force report, 1885, 1916
force report definition, 1893
Force Reports dialog box, 601, 1916, 2614
forces
coefficients of, 1916, 1924
exporting data, 69
on boundaries, 1916
formation enthalpy, 528, 987
formation rate, 1982
Fourier Transform dialog box, 1875, 2602
Fractional Step algorithm, 1562
free stream, 347
freezing the temperature, 297
friction coefficient, 1305
friction collision law, 1257
friction packing limit, 1450, 1460
frictional modulus, 1450, 1460
frictional pressure, 1450, 1459
frictional viscosity, 1459
frozen flux formulation, 1612
fuel cell, 277
fuel NOx parameters, 1160
fuel streams
NOx model, 1156
SOx model, 1173
full multicomponent diffusion, 520
full multigrid (FMG), 1595
convergence strategies, 1597
using, 1596
fully-developed flow, 582
Fuse Face Zones dialog box, 212, 2708
fusing face zones, 212
FW-H acoustics model, 1207

G

G-equation, 1092
GAMBIT files
importing, 64
meshes, 136
pathlines, 1778
reading, 44
gas constant, definition, 1973

gas law, 479, 484, 486, 530, 596
gaseous and liquid fuel NOx parameters, 1160
gauge pressure, 531
general non-reflecting boundary conditions
  overview of, 416
  theory, 416
  using, 421, 424
General task page, 590, 1553, 2160
Generic Erosion Model Parameters dialog box, 1309
GeoMesh mesh files, 136
  importing, 64
  reading, 44
geometric reconstruction scheme, 1373
geometric roughness height, definition
  definition, 1973
Geometry Based Adaption Controls dialog box, 2709
Geometry Based Adaption dialog box, 1699, 2709
geometry-based adaption, 149, 1698
global matrix size, 2056
glossary, 2871
gnuplot, 1150
governing equations
  source term additions to, 297
GPGPUs
  algebraic multigrid (AMG), 2056
  discrete ordinates (DO) radiation model, 2065
  view factors, 2064
gradient adaption, 1688
  dynamic, 1690
Gradient Adaption dialog box, 1688, 2710
gradient option
  setting, 1553
granular bulk viscosity, 1459
granular conductivity, 1460
  definition, 1973
granular temperature, 1450, 1460
granular viscosity, 1450, 1459
graphical user interface (GUI), 1
  customizing the, 22
graphics, 1747
  overlay, 1781
  overlaying, 1785
  picture files, 102
  scene, 1781
  window dumps, 106
Graphics and Animations
  contours, 1757
  pathlines, 1772
  sweep surface, 1782
  vectors, 1765
Graphics and Animations task page, 2554
graphics device information, 1800
Graphics Periodicity dialog box, 1813, 2586
graphics windows, 3
  active, 1787
  captions, 1788
  closing, 1786
  hiding, 1783
  multiple, 1786
  opening, 1786
  titles
    adding, 1788
    visibility, 1788
  Windows features, 5
gravitational acceleration, 835
Gray-Band Absorption Coefficient dialog box, 2349
Gray-Band Refractive Index dialog box, 2350
grid
  case check, 1657
  coarse levels, 1576
grid resolution RANS turbulence models, 756
Grid Resolution SRS turbulence models, 760
  free shear flows, 760
  wall boundary layers, 760
GT-POWER, 451
GUI, 1
  customizing the, 22

H

hanging nodes / edges, 115, 205
  effect on polyhedral conversion, 203-205
  removing, 143
harmonic response analysis, 1222
HCN density, definition, 1974
HDF (hierarchical data format), 47
  overset cases, 191
heat capacity, 513
heat exchanger groups, 920
Heat Exchanger Model dialog box, 932, 943, 2200
heat exchanger models, 396, 919
  choosing, 920
  dual cell, 920-921
    inputs, 922
    restrictions, 922
  effectiveness, 941
  macro model, 920
    features, 930
    grouped, 943
    restrictions, 931
    ungrouped, 932
  macros, 938
  NTU model, 920
    restrictions, 931
  number-of-transfer-units (NTU) model

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

features, 930
postprocessing, 950
reports, 950
restrictions, 920
simple effectiveness model, 920
features, 930
restrictions, 931
Heat Exchanger Report dialog box, 2625
heat flux, 384, 1911
diffusive, 373
exporting data, 832
total, 1997
wall, 383
heat of formation, 528
heat of reaction, 983
definition, 1974
heterogeneous, 1974
surface reaction, 1000
heat sources, 260, 297, 299, 396
heat transfer, 825
augmented, 373
boundary conditions, 384
at walls, 370
buoyancy-driven flows, 833
convective and conductive, 825
inputs for, 825
natural convection flows, 833
overview of models, 825
periodic, 912
postprocessing, 831
radiation, 384, 845
rate, 1911
reporting, 831
averaged coefficients, 832
for radiation, 882
through boundaries, 832
through surface, 832
shell conduction, 376, 838
initialization, 1593
limitations, 839
locking the temperature, 297
managing, 840
physical treatment, 838
postprocessing, 843
with the S2S radiation model, 851
solution
procedures, 825, 828
strategies, 828
to the wall, 382
volume sources, 260, 297
heat transfer coefficient, 133, 1924
calculations of, 384
definition, 1995
Heat Transfer Data Table dialog box, 933, 2204
helicity, definition, 1974
Help menu
PDF, 25
help system, 23
text user interface, 36
Herschel-Bulkley dialog box, 496, 2326
Herschel-Bulkley model, 495
heterogeneous reactions
defining, 1380
hex cells, 127
hexcore meshes
limitations, 203-205
removing hanging nodes / edges, 205
removing hanging nodes / edges from, 143
hidden line removal, 1798
hidden surface removal, 1798
hiding graphics windows, 1783
hierarchical data format (HDF), 47
overset cases, 191
high order term relaxation, 1557
Histogram dialog box, 1851, 1924, 2598
histogram plots
options, 1852
histograms, 1924
plots, 1839, 1851
reports, 1924
hybrid initialization, 1597
solution strategies, 1599
using, 1597
Hybrid Initialization dialog box , 2524
hybrid RANS-LES turbulence models, 757
hydraulic diameter, 304
HYPERMESH ASCII files
importing, 65

I

I-deas Universal files, 137
exporting, 69, 81
during a transient calculation, 86
importing, 65
mapping data with, 97
ICEM CFD mesh file, 137
ideal gas law, 347, 484, 486, 530, 596, 1268
ignition source, 992
impedance boundary condition, 405
impedance boundary conditions, 422
impeller-baffle interaction, 613
implicit formulation
convergence acceleration
stretched meshes, 1573

implicit interpolation scheme, 1373
implicit update option, 643
  settings, 709
Import Particle Data dialog box, 2713
import surfaces, 1714
importing
  ABAQUS files, 62
  ANSYS FIDAP files, 64
  CFX files, 62
  CGNS files, 63
  CHEMKIN files, 68, 1202
  EnSight files, 64
  fe2ram filter, 142
  files, 59
  GAMBIT files, 64
  GeoMesh files, 64
  HYPERMESH ASCII files , 65
  I-deas Universal files, 65
  LSTC files, 65
  Marc POST files, 65
  Mechanical APDL files, 66
  meshes, 135
  NASTRAN files, 66
  particle data, 1338
  PATRAN files, 67
  PLOT3D files, 67
  PTC Mechanica Design files, 67
  Tecplot files, 67
Imprint Surface dialog box, 2714
Improve Mesh dialog box, 231, 2715
improving meshes, 194
in cylinder combustion, 1112
in-cylinder model
  crevice model
    output file, 1150
    postprocessing, 1148
in-cylinder option, 643, 695
  events, 716
  geometry and motion attributes, 699
  mesh topology, 696
  piston pin offset, 690
  settings, 690
  valve opening and closing, 705
In-Cylinder Output Controls dialog box, 2481
incident radiation
  definition, 1974, 1995
  reporting, 885
incomplete particle trajectories, 1247
incompressible flow calculations at inlet pressure boundaries, 315
Inert dialog box , 2249
inert EGR, 1077, 1112
inert model, 1075
  EGR
    resetting, 1077
inertial resistance
  alternative formulation for, 270
inflation layers, 681, 683
  deforming zones, 734
Inflation Settings dialog box, 723, 734, 2498
information
  graphics device, 1800
initial conditions, 1591
  discrete phase, 1265
  Eulerian wall film model, 1542
initialization
  hybrid, 1597
initializing the solution, 1591
  using average values, 1592
injections
  copying, 1286
  creating, 1285
  deleting, 1286
  listing, 1286
  modifying, 1285
  reading, 1286
  writing, 1286
Injections dialog box, 1284, 2716
inlet boundaries, 316
inlet vent boundary, 302, 334
Inlet Vent dialog box, 334, 2400
Input Parameter Properties dialog box, 2631
input parameters, 2125
  advanced, 247
  use in Scheme procedure, 247
  use in UDF, 249
input signal, 1877
Input Summary dialog box, 1927, 2717
input summary report
  generating a, 1927
inputs
  shallow wave, 1424
  wave group, 1425
  wave spectrum, 1425
intake fan boundary, 302, 336
Intake Fan dialog box, 336, 2406
integral reporting, 1920, 1922
integration parameters
  setting, 1128
Integration Parameters dialog box, 1117-1118, 2235
interconnects, 2009
  Linux, 2028
  Windows, 2023
interface

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

mapped
creating, 164
heat transfer, 378
interface option, 158
interface boundary metrics, 162
Interface dialog box, 2411
interface modeling options, 1371
Interface Options dialog box, 2717
interface zone, 164, 630, 634
contour display, 639
interfaces
non-conformal, 150
interfaces, non-conformal
,matching, 157
coupled, 156
mapped, 158
periodic boundary condition, 153
periodic repeats, 154
setting up, 164
static, 160
Interior Cell Zone Selection dialog box, 1214, 2294
Interior dialog box, 2411
interior zone, 76, 237, 391, 399, 2411
intermittency effective, definition, 1975
intermittency factor, definition, 1975
intermittency transition model, 800
internal emissivity, 871, 875
internal energy, definition, 1975
Interpolate Data dialog box, 94, 2652
interpolation, 94
file format, 96
for fluid-structure interaction (FSI) problems, 97
mesh-to-mesh, 94
Interpreted UDFs dialog box, 2719
interrupt
of calculations, 1601
interrupts , 34
Intrinsic Combustion Model dialog box, 2353-2354
inviscid flows, 599
inputs for, 600
solution strategies for, 600
ISAT, 1128
dimension reduction, 1128, 1136
Iso-Clip dialog box, 1730, 2720
Iso-Surface dialog box, 1728, 2721
Iso-Value Adaption dialog box, 1692, 2722
Iso-Value dialog box, 1821, 2581
isosurfaces, 1709, 1728
isotropic diffusion UDS, 508
isotropic scattering phase function, 518
isovalue, 1729-1730
adaption, 1691
modification, 1821
iteration, 1600
appending to file name, 48
control during, 1638
interrupt of, 1601
iterative procedure, 1600

J

jet acoustic power, definition, 1975
Joule heat source, definition, 1975
journal files, 57
jump
adhesion
inputs for, 1394, 1442

K

k-epsilon model, 752
curvature correction, 798
model compatibility, 2847
k-epsilon multiphase turbulent viscosity
customizing, 1478
k-omega model, 753
curvature correction, 798
intermittency transition model, 800
model compatibility, 2847
key frame, 1825
kinetic energy
subtest, 1994
kinetic reaction rate, definition, 1975
kinetic theory, 490-491, 499, 515, 526
parameters, 529
Kinetics/Diffusion-Limited Combustion Model dialog box, 2352, 2354
kuhnke wall model, 1303, 1542

L

labels, 2608
colormap, 1793
Lagrangian discrete phase model, 1235
Lagrangian dispersed phase model
cubic equation of state model, 548
laminar finite-rate model, model compatibility, 2847
laminar flow shear stress, 382
laminar kinetic energy, definition, 1976
laminar viscosity, 1979
laminar zone, 255, 283
laminar-turbulent transition models, 754
Laplace coarsening, 1580
Laplace smoothing, 2033
Laplacian smoothing, 231, 657
large eddy simulation (LES)
reporting mean flow quantities, 822

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

large eddy simulation (LES) turbulence model, 757
  model compatibility, 2847
  solution strategies, 813
latent heat, 1527
  droplet temperature, 1252
  of vaporization, 1049, 1322
layering
  dynamic, 661
LEE noise source, definition, 1976
legend, 1788, 1845
length scale, 304
Lennard-Jones parameters, 490, 526, 529
LES subgrid turbulent viscosity, definition, 1976
LES turbulence model
  model compatibility, 2847
  see also large eddy simulation (LES) turbulence model, 757
Lewis number, 1000
license users, 26
lift coefficient, 1916
lift force, 1251
lift report definition, 1893
Lift Report Definition dialog box, 2724
light-off, 277
lights, 1796, 1818
Lights dialog box, 1797, 2588
Lilley's noise source, definition, 1976
limiter, 1590
  filter, 1590
line surfaces, 1709, 1718
  using the line tool, 1720
line tool, 1719-1720
Line/Rake Surface dialog box, 1718, 2727
linear-anisotropic scattering phase function, 518
linearize source terms, 1264
linearly elastic solid based smoothing method, 656
  applicability, 656
liquid fraction, definition, 1976
liquid fuel combustion, 1032
  in non-premixed combustion model, 1045
liquidus temperature, 1527
load balancing, 2033
load distribution
  computation, 893
local cell remeshing method, 669
local cylindrical coordinate system, 311
local cylindrical velocities, 318
local face remeshing method, 669
local remeshing
  additional, 670
  method, 668
locking the temperature, 297
log files, 57
logarithmic plots, 1854
look-up tables
  NIST, 555
    binary saturation tables, 558
loss coefficient, 396
  core porosity model, 941
  inlet vent, 335
  outlet vent, 355
  radiator, 399
LS-DYNA, 65
LSF
  job scheduling, 2015
LSTC files
  importing, 65
lumped parameter models, 389

M

Mach number, 348, 594, 599
  definition, 1977
Macro Heat Exchanger Group dialog box, 2212
macro heat exchanger model, 920, 930
  grouped, 920
    inputs, 943
  restrictions, 931
  ungrouped, 920
    inputs, 932
macro heat exchanger models, 930
macros, 938, 1640
  during mesh morpher/optimizer runs, 2125
magnifying the display, 1800, 1807
Magnus lift law
  none, 1291
  Oesterle-Bui-Dinh, 1291
  Rubinow-Keller, 1291
  Tsuji-et-al, 1291
Manage Adaption Registers dialog box, 1701, 2728
Manage Adjoint Observables dialog box, 2084
mapped mesh interface
  heat transfer, 378
  interface option, 158
mapping data
  fluid-structure interaction (FSI) problems, 97
Marangoni stress, 361, 363
Marc POST files
  importing, 65
marking poor elements, 194, 1977
mass diffusion
  to surfaces, 1000
mass diffusion coefficients, 519, 527, 986, 1969, 1976
  about, 519
  full multicomponent, 520

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

inputs for, 524
kinetic theory, 526
using Fick's Law, 519
Mass Diffusion Coefficients dialog box, 525-526, 2344
mass flow inlet boundary, 302, 322
limitations, 323
special considerations, 323
mass flow outlet boundary, 330
limitations, 330
Mass Flow Outlet dialog box, 2417
mass flow rate, 1911
defining for an inlet, 324
defining for an outlet, 331
reporting for Eulerian wall film, 1545
transport equations, 574
mass flow split, 352
mass flux, 1911
defining for an inlet, 324
defining for an outlet, 331
mass fraction, 1977
mass source terms
defining, 299
mass sources, 299
Mass-Flow Inlet dialog box, 323, 2411
Mass-Flow Outlet dialog box, 330
matching wall interface option, 157
material properties
checking, 1664
Material Properties dialog box, 2315
materials, 460
copying from the database, 464
creating new, 466
database, 460, 959, 975
data sources, 464
user-defined, 467
deleting, 466
discrete phase, 461, 1288, 1313, 1316
mixture, 461, 959
modifying, 463
PDF mixture, 1080
renaming, 463
reordering, 467
saving, 466
Materials task page, 461, 508, 2300
matrix size
global, 2056
Maxwell-Stefan equations, 520
McLaury Model Parameters dialog box, 1310
mean
custom field functions, 1623, 1978
flow quantities, 1623
solution variable, 1978
mean beam length, 516
mean mixture fraction, 300
Mechanical APDL files
exporting, 69
after a calculation, 72
during a transient calculation, 73
importing, 66, 140
mapping data with, 97
Mechanical APDL Input, 69
Mechanical APDL Input files
exporting, 69, 73
during a transient calculation, 86
Mechanism dialog box , 2270
memory usage, 198
in multigrid, 1576, 1580
menu commands
character strings, 35
Merge Zones dialog box, 206, 2730
merging zones, 206
meridional view, 1820
mesh
adaption, 1681
coarsening
based on gradient, 1689
near walls, 1695
filters
partition, 2052
tpoly , 143
interfaces, 630
creating, 616, 634
shapes of, 631
motion of, 603, 628
moving reference frames, 603
partitioning, 2030
automatic, 2031
check, 2052
filter, 2052
guidelines, 2033
inputs for, 2031, 2033
interpreting statistics, 2052
manual, 2033
methods, 2046
METIS, 2052
optimizations, 2050
report, 2043
statistics, 2043
troubleshooting, 2056
within registers, 2043
within zones, 2043
polyhedra
limitations, 1684
refinement, 297, 1681, 1684

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

anisotropic, 1697
at boundaries, 1685
based on cell volume, 1694
based on gradient, 1688
based on isovalue, 1691
dynamic, 1690
in a region, 1692
near walls, 1695
replacement, 222
requirements
dynamic meshes, 649, 664, 669, 680, 683, 686, 690
moving reference frame, 605
multiple reference frames, 616
rotating flow, 589
sliding meshes, 633
swirling flow, 589
resolution, 1681, 1684
setup constraints
dynamic meshes, 649, 664, 669, 680, 683, 686, 690
moving reference frame, 605
multiple reference frames, 616
rotating flow, 589
sliding meshes, 633
swirling flow, 589
setup time, 1681
smoothing, 230
spacing at walls
in laminar flows, 382
in turbulent flows, 1695
units of, 112
velocity, 1978
Mesh Adaption Controls dialog box, 1705, 2730
Mesh Colors dialog box, 2168
Mesh Display dialog box, 1750, 2163
Mesh Interfaces dialog box, 164, 616, 634, 2732
Mesh Method Settings dialog box, 2474
mesh morpher/optimizer, 2122-2123
compass optimizer, 2123
file formats, 2867
limitations, 2122
newuoa optimizer, 2123
powell optimizer, 2124
process, 2122
rosenbrock optimizer, 2125
setting up, 2125
simplex optimizer, 2124
torczon optimizer, 2124
Mesh Morpher/Optimizer dialog box, 2125, 2734
Mesh Motion dialog box, 743, 2500
mesh motion methods, 644
feature detection, 690
volume mesh update, 689
Mesh Scale Info dialog box, 665, 2477
mesh/check-verbosity , 133, 193
mesh/modify-zones/copy-move-cell-zone , 222, 922
mesh/modify-zones/extrude-face-zone-delta , 203-205, 217
mesh/modify-zones/extrude-face-zone-para , 203-205, 218
mesh/modify-zones/fuse-face-zones , 213-214
mesh/modify-zones/make-periodic , 214
mesh/modify-zones/matching-tolerance , 214
mesh/modify-zones/mrf-to-sliding-mesh , 634
mesh/modify-zones/orient-face-zone , 217
mesh/modify-zones/slit-face-zone , 216
mesh/modify-zones/slit-periodic , 215
mesh/polyhedra/convert-hanging-nodes , 205
mesh/polyhedra/options/preserve-interior-zones , 203
mesh/repair-improve/allow-repair-at-boundaries , 194
mesh/repair-improve/improve-quality , 194
mesh/repair-improve/include-local-polyhedra-conversion-in-repair , 194
mesh/repair-improve/repair-face-handedness , 194
mesh/repair-improve/repair-face-node-order , 194
mesh/repair-improve/repair-periodic , 194
mesh/repair-improve/report-poor-elements , 194
mesh/smooth-mesh, 231, 234
mesh/swap-mesh-faces, 235
meshes, 115
accuracy
cell quality, 134
flow-field dependency, 135
smoothness, 135
activating zones, 221
adaption, 134
aspect ratio, 129, 132, 134
C-type, 214
checking, 192
face handedness, 192
face node order, 192
polyhedral cells, 192
quality, 133
chimera, 172
choosing type of, 128
colors when displaying, 1752
computational expense, 129
connectivity
face-node, 121
converting to polyhedra, 199
preserving interior surfaces, 203
copying zones, 222
creating, 44, 135-136
deactivating zones, 220
deleting zones, 219

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

displaying, 1748
extrusion, 217
face-node
  hex cells, 127
  polyhedral cells, 128
  pyramidal cells, 126
  quadrilateral cells, 123
  tetrahedral cells, 124
  triangular cells, 122
  wedge cells, 125
files, 44
  reading multiple, 145, 212
filters
  fe2ram , 142
  fl42seg , 136
  tmerge , 148
hexcore, 203-205
importing, 135
  ANSYS FIDAP files, 145
  ANSYS Meshing files, 135
  CFX files, 141
  CGNS, 63
  FLUENT 4 files, 144
  Fluent Meshing files, 136
  Fluent meshing mode files, 136
  Fluent/ UNS files, 144
  GAMBIT files, 136
  GeoMesh files, 136
  I-DEAS files, 137
  ICEM CFD files, 137
  Mechanical APDL files, 140
  NASTRAN files, 138
  PATRAN files, 139
  PreBFC files, 136
  RAMPANT files, 144
interfaces, 150
  creating, 164
  deleting, 164
  setting up, 226
manipulating, 115
mapped
  heat transfer, 378
  setting up, 164
memory usage, 198
modifying zones, 218
multiblock, 145, 212
non-conformal, 150
  algorithm, 162
  requirements, 163
  setting up, 164
O-type, 214
orthogonal quality, 131, 134, 1980
overset, 172
partitioning
  METIS, 68
  model-weighted, 2039
  report, 199
  statistics, 199
periodic repeats interface option, 154, 164
periodicity
  conformal, 214
  non-conformal, 153
polyhedra
  adaption, 203
  advantages, 199
  converting cells with hanging nodes / edges to, 143, 205
  converting domain to, 199
  converting skewed cells to, 204
  face-node connectivity, 128
  hanging nodes / edges, 203-205
  limitations, 203-205
quality
  accuracy, 131
  aspect ratio, 132
  element distribution, 133
  improving, 194
  mesh morpher/optimizer, 2125
  orthogonal quality, 131
  stability, 131
reading, 44, 115, 135
recommendations, 129
reordering, 226
repairing, 194
replacing, 222
replacing zones, 218
requirements, 130
  axisymmetric, 130
  non-conformal, 163
  periodic, 130
resolution, 129, 133
rotating, 229
scaling, 226
setup constraints, 130
  axisymmetric, 130
  non-conformal, 163
  periodic, 130
setup time, 128
size, 197
skewness, 129, 134
spacing at walls, 133
statistics
  reporting of, 197
topologies, 115

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

examples of, 116
hexahedral, 115
polyhedral, 115
quadrilateral, 115
tetrahedral, 115
triangular, 115
translation, 228
types of, 115
which types to use, 129
zone information, 199
meshing mode, 1
mesh files, 136
reading, 44
reading multiple mesh files, 147
Meshing task page , 2159
message passing library, 2009
method of moments soot formation model, 1196
METIS, 68, 2052
Microsoft Job Scheduler, 2023
job scheduling, 2015
MiniVAS Settings dialog box, 2742
Mirror Planes dialog box, 1815, 2585
mirroring the domain, 1811
mixed convection, 833, 1564
inputs for, 834
mixing plane model, 615
enthalpy conservation, 623
fixing the pressure level, 622
model compatibility, 2847
patching values, 1594
pressure boundary conditions in, 619
setup, 619
solution initialization, 1592, 1598
solution procedures, 624
swirl conservation, 623
velocity formulation in, 619
with non-reflecting boundary conditions, 415
Mixing Planes dialog box, 619, 2744
mixing rate, 961, 982, 1380
mixture diffusivity, 574
mixture fraction
boundary conditions, 1079
secondary, 1988
variance, 1978
secondary, 1988
mixture materials, 461, 959
creating, 975
PDF, 1080
mixture multiphase model
body forces, 1377
droplet diameter, 1448
model compatibility, 2847
properties, 1447
solution method
coupled pressure-based solver, 1507
solution strategies, 1516
mixture properties, 972, 986
mode
meshing, 1
mesh files, 44, 136
reading multiple mesh files, 147
solution, 1
reading multiple mesh files, 146
model selections
checking, 1659
model-weighted partitioning, 2039
Models
Energy..., 825
Heat Exchanger
Dual Cell Model, 922
Multiphase, 1367
Radiation, 845, 851, 865, 879
Solidification and Melting, 1527
Species, 1091
Models task page , 2169
modified turbulent viscosity
definition, 1978
modifying zones, 218
molar concentration of species
definition, 1978
mole fraction
soot, 1979
species, 1978
molecular viscosity, definition, 1979
molecular weight, 484-485, 987
moment report, 1916
moment report definition, 1893
Moment Report Definition dialog box, 2746
moments
coefficients of, 1916, 1924
reporting, 1916
momentum accommodation coefficient, 987
momentum source terms
defining, 299
momentum sources, 299
momentum thickness Re, definition, 1979
monitoring
convergence, 1635
forces and moments, 1612
optimization history for the mesh morpher/optimizer, 2125
residuals, 1624
solution convergence, 1624
statistics, 1633

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

surfaces, 1612
Monte Carlo (MC) radiation model
boundary conditions, 876
inputs for, 867
solution parameters, 880
morpher, 2067
file formats, 2867
Moss-Brookes soot formation model, 1190
Moss-Brookes-Hall soot formation model, 1190
Motion Settings dialog box, 2125, 2749
mouse
functions, 1800
GAMBIT-style, 1800
manipulation, 1798, 1800
probe, 1800
moving domains, model compatibility, 2847
moving mesh, 603
moving reference frame, 613
boundary conditions, 608
constraints, 609
coordinate-system constraints, 605
postprocessing, 611, 1766
pressure boundary conditions, 608
setup, 606
single, 605
solution procedures, 610
velocity formulation , 606, 609
moving reference frames, 603
patching values, 1594
postprocessing, 1766
solution initialization, 1592, 1598
moving walls, 359
MPEG file, 1653, 1829
MSC Marc, 65
mud, 496
multi-stage scheme
controls for, 1588
modifying, 1588
stability, 1571
multiblock meshes, 145, 212
multicomponent diffusion, 520
theory, 520
multicomponent diffusion model, 519
multigrid solver
aggregative
bi-conjugate gradient stabilized method (BCG-STAB), 1578
conjugate gradient method (CG), 1578
recursive projection method (RPM), 1578
algebraic (AMG)
inputs for, 1577
coarsening parameters, 1580
default parameters, 1583
flexible cycle, 1577
full (FMG), 1595
convergence strategies, 1597
using, 1596
full-approximation storage (FAS)
creating coarse grid levels for, 1576
inputs for, 1575, 1583
levels, 1576
inputs for
algebraic (AMG), 1577
full-approximation storage (FAS), 1575, 1583
memory usage, 1576, 1580
performance monitoring, 1577
residual reduction tolerance, 1578
selective, 1578
termination criteria, 1578
turning off, 1580, 1583
V cycle, 1577
W cycle, 1577
with parallel solver, 1577
multiphase flows
body forces, 1377
boundary conditions, 1391
case check, 1519
compressible, 1375
heterogeneous reactions, 1380
inputs for, 1365
cavitation model, 1455
heat transfer in Eulerian model, 1478
interfacial area in Eulerian model, 1480
mass transfer, 1383
turbulence interaction in Eulerian model, 1476
model compatibility, 2847
postprocessing, 1520
dense discrete phase model, 1523
wet steam flow, 1523
reporting flow rates, 1524
solution strategies, 1504
solving wet steam flow, 1518
user-defined scalar transport equations, 581
using the wet steam model, 1495
wet steam model
properties, 1495
UDF, 1496
multiphase model
coupled solution, 1504, 1506
patching initial volume fraction, 1407
Multiphase Model dialog box, 1367, 2171
multiphase species transport, 1378
postprocessing, 1522
multiple graphics windows, 1786

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

multiple reference frames, 612
  boundary conditions, 616
  discrete phase in , 614, 1238
  for axisymmetric flow, 614
  mesh setup, 616
  patching values, 1594
  pathlines, 614
  postprocessing, 624, 1766, 1927
  pressure boundary conditions in, 616
  restrictions of, 613-614
  setup, 616
  solution initialization, 1592, 1598
  solution strategies, 624
  steady flow approximation, 613
  velocity formulation in, 614, 616
multiple reference frames, model compatibility , 2847
multiple surface reactions model
  dialog box, 2354
  using, 1002

N

N2O density, 1979
naming files
  options for, 43
NASTRAN files, 138
  exporting, 69, 82
    during a transient calculation, 86
  importing, 66
  mapping data with, 97
natural convection, 833, 1564
  high-Rayleigh-number flows, 837
  inputs for, 834
  modeling, 833
  operating density, 836
  solution procedure, 834
natural time, 495
near-wall treatments, 767, 769
  Werner-Wengle, 792
New Material Name dialog box, 469, 2317
New Report File, 1904
New Report File dialog box, 2754
New Report Plot, 1906
New Report Plot dialog box, 2755
newuoa optimizer, 2123
NH3 density, 1979
NIST real gas model, 532, 550
  limitations, 551
NO density, 1979
no-slip condition, 358, 361-362
node based averaging
  discrete phase, 1263
node values
  flow variables, 1931
  for contours, 1761
  for pathlines, 1780
  for XY plots, 1844
nodes
  display of, 1752, 1798
  precision of coordinates, 46
noise, 1206
nomenclature, 2871
non-blocking surfaces, 858
non-condensable gases, 1455
non-conformal interface algorithm, 162
non-conformal meshes, 150
  requirements for, 163
non-equilibrium chemistry
  rich flammability limit, 1035
non-gray discrete ordinates (DO) radiation model
  absorption coefficient, 869
  black body emission factor, 865
  emissivity weighting factor, 865
  inputs for, 865
  refractive index, 869
non-gray P-1 radiation model
  absorption coefficient, 869
  black body emission factor, 847
  emissivity weighting factor, 847
  inputs for, 847
  refractive index, 869
non-gray radiation model
  absorption coefficient, 869
  discrete ordinates (DO) radiation model
    inputs for, 865
  DO/energy coupling, 866
  P-1 radiation model
    inputs for, 847
  refractive index, 869
non-isotropic thermal conductivity
  user-defined, 277
non-iterative time advancement (NITA), 1612
  inputs, 1566
  supported models, 1568
non-Newtonian fluids, 492
non-Newtonian power law, 493
Non-Newtonian Power Law dialog box, 493, 2324
non-Newtonian viscosity, 492
  Carreau model, 494
  Cross model, 495
  Herschel-Bulkley model, 495
  power law, 493
  temperature dependent, 492
non-premixed combustion model

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS ANSYS Fluent solution secondary mixture fraction parameters, 1084
boundary conditions, 1079
condensed phase species, 1045
empirical fuels, 1034, 1045
equilibrium chemistry, 1025, 1031
flamelet data, 1060
for coal combustion, 1032, 1044, 1046
for liquid fuels, 1032
fuel inlet temperature, 1044
look-up tables, 1063, 1068
look-up tables with Flamelet Generated Manifolds, 1107
model compatibility, 2847
non-adiabatic form, 1032
non-equilibrium chemistry
rich flammability limit, 1035
postprocessing, 1085
problem setup, 1026
secondary stream, 1033
solution parameters, 1084
solution parameters, 1063
solving, 1083
species selection, 1044
stability issues, 1068
non-reacting species transport, 1011
non-reflecting boundary conditions, 344, 405
general, 416
limitations of, 406, 416
parallel processing with, 415
target mass flow rate, 344
turbo-specific, 406
with mixing plane model, 415
normalization, 1924
of residuals, 1625-1626, 1628
normalized concentration of nuclei, definition, 1979
normalized Q criterion, definition, 1979
normalized soot aggregation moments, definition, 1980
normalized soot moments, definition, 1980
NOx model, 1053, 1153
boundary conditions, 1169
coal, 1160
fuel, 1154
fuel streams, 1156
gaseous fuel, 1160
inputs for, 1153
intermediate N2O, 1154
liquid fuel, 1160
model compatibility, 2847
prompt, 1154
thermal, 1154
under-relaxation for, 1169
user-defined functions for, 1158
NOx Model dialog box, 1154, 2250
NTU model, 920
restrictions, 931
NTU Table dialog box, 2205
number-of-transfer-units (NTU) model
features, 930
numbering files, 43
numerical beach
open channel, 1432
open channel wave boundary condition, 1432
numerical diffusion, 129, 1555
Nusselt number, 373
definition, 1995

O

O-type meshes, 214
Objective Function Definition dialog box, 2125, 2757
objective functions, 2122-2123, 2125
defining, 2757
observables for the adjoint solver, 2069, 2084
arithmetic average, 2089
creating, 2085
editing, 2087
fixed value, 2069, 2088
force, 2069, 2088
linear combination, 2089
mean variance, 2089
moment of force, 2069, 2088
product, 2089
ratio, 2089
surface integral, 2070, 2088
swirl, 2069, 2088
total pressure drop, 2069, 2088
unary operation, 2089
Oesterle-Bui-Dinh, 1291
oil-flow pathlines, 1776
Oka Model Parameters dialog box, 1311
one DOF motion, 707
one-step soot formation model, 1185
online help for the GUI, 23
Online Technical Resources
access from the interface, 26
opaque walls, 871
open channel
random wave boundary, 1419
recommendations, 1429
open channel boundary condition, 1410
limitations, 1417
recommendations, 1418
open channel flow, 1410
limitations, 1417

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

recommendations, 1418
open channel wave boundary condition, 1419
Open Database dialog box, 2313
Operating Conditions dialog box, 484-485, 531, 596, 834, 2382
operating density, 836
operating pressure, 484-485, 530, 596, 835
floating, 597
operating temperature, 835
operations for the adjoint solver, 2072
arithmetic average, 2073
linear combination, 2073
mean variance, 2073
product, 2073
ratio, 2073
unary, 2073
optical thickness
energy coupling for non-gray DO radiation, 866
Optimization History Monitor dialog box, 2125, 2759
optimizer, 2067
file formats, 2867
Options dialog box, 2478
options for startup
parallel
Linux, 2026
Windows, 2021
Orient Profile, 2387
Orient Profile dialog box, 443
Orientation Calculator dialog box, 2497
orienting
face zones, 217
pictures, 105
profiles, 443
orthogonal quality, 131, 134
definition, 1980
mesh morpher/optimizer, 2125
orthographic, 1808
Orthotropic Conductivity dialog box, 503, 2330
orthotropic diffusion UDS, 511
orthotropic thermal conductivity, 503-504
Orthotropic UDS Diffusivity dialog box, 511
outflow boundary, 302, 350
Outflow dialog box , 2420
outflow gauge pressure, 320
outlet vent boundary, 302, 353
Outlet Vent dialog box, 353, 2422
outline display, 1748, 1753
output parameters, 1909, 2125, 2627, 2757
overlaying graphics, 1785
Overset Interfaces task page, 2470
overset meshes, 172

P

P-1 radiation model
material properties, 868
non-gray
absorption coefficient, 869
black body emission factor, 847
emissivity weighting factor, 847
inputs for, 847
refractive index, 869
properties, 515, 876
residuals, 881
solution parameters, 878
P-1 radiation source terms, 300
packed beds, 261, 274
packing limit, 1451, 1461
parallel
GPGPUs, 2056
parallel check, 2060
Parallel Connectivity dialog box, 2059, 2760
parallel processing, 2009
active cell partition, 1959
architecture, 2009
automatic partitioning, 2031
cell partition, 1962
communication, 2064
compute cluster package (CCP), 2023
data files, 51
efficiency, 2009
exporting, 69
input/output (I/O) capability, 51
introduction, 2009
launching Fluent
on Linux, 2026
on Windows, 2021
using Linux command line options, 2026
using the Fluent Launcher, 2012
using the Microsoft Job Scheduler, 2023
using Windows command line options, 2021
limitations, 69, 860
load balancing, 2044
load distribution, 2055
manual partitioning, 2033
model-weighted partitioning, 2039
multigrid settings for, 1577
network connectivity, 2059
network latency and bandwidth, 2062
on a Linux system
using command line options, 2026
on a Windows system
using command line options, 2021
using the Fluent Launcher, 2012
using the Microsoft Job Scheduler, 2023

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

partition surfaces, 1712
partitioning, 2009, 2030
model-weighted, 2039
troubleshooting, 2056
performance, 2060, 2064
clearing the file cache buffers, 2066
statistics, 2060
recommended procedure, 2009
stored cell partition, 1993
thread control, 2058
parallel/partition/set/dpm-load-balancing , 1360
parallel/partition/set/load-distribution , 2055
Parameter Bounds dialog box, 2125, 2760
parameters, 1909
defining, 243
fuel NOx, 1160
gaseous and liquid fuel NOx, 1160
input, 243
prompt NOx, 1159
solid (coal) fuel NOx, 1161
SOx, 1175
turbulence, 1166, 1179
Parameters and Customization task page , 2632
Parameters dialog box, 2627
partially premixed combustion model, 1099
in cylinder, 1112
limitations of, 1099
look-up tables, 1105
model compatibility, 2847
overview, 1099
solution parameters, 1105
using, 1100
partially premixed flames, 1089-1090, 1099
Participating Boundary Zones dialog box, 2197
particle, 1235
accretion, 1251, 1309
cloud tracking, 1296
condensation, 1307
erosion, 1251, 1309
incomplete, 1247
initial conditions, 1265
laws
custom, 1288, 1296
radiation, 1324
reference frame, 1263
rotation, 1305
size distribution, 1265, 1281
trajectory calculations, 1240
trajectory reports, 1340, 1347
alphanumeric reporting, 1351
display of, 1330
PDF tracking, 1124
step-by-step track report, 1347
unsteady, 1349
turbulence, 1252
wall boundary layer model, 1309
particle data
importing, 1338
Particle Filter Attributes dialog box, 2561
particle history data
exporting
steady-state, 84
transient, 86
particle rotation
discrete phase, 1291
friction coefficient, 1305
particle rotations
limitations, 1238
Particle Sphere Style Attributes dialog box, 1332, 2563
particle summary
reporting, 1358
Particle Summary dialog box, 1359, 2625
particle surface reactions, 974, 977, 1001
catalyst species, 1002
inputs for, 1001
particle tracking, 1240
particle tracks, 1332
Particle Tracks dialog box, 1330, 2761
Particle Vector Style Attributes dialog box, 1334, 2564
partition boundary cell distance, definition, 1980
partition neighbors, 2046
partition neighbors, definition, 1980
Partition Surface dialog box, 1712, 2766
partition surfaces, 1709, 1712
partitioning, 68, 1962, 1980, 2009, 2030
automatic, 2031
check, 2052
filter, 2052
guidelines, 2033
inputs for, 2031, 2033
manual, 2033
methods, 2046
METIS, 68, 2052
model-weighted, 2039
optimizations, 2050
report, 199, 2043
statistics, 199, 2043
interpreting, 2052
troubleshooting, 2056
using dynamic mesh model, 2039
within registers, 2043
within zones, 2043
Partitioning and Load Balancing dialog box, 2033, 2767
passage loss coefficient, 1863

Patch dialog box, 1593, 2522
Patch Reconstructed Interface, 1407
patching
  field functions, 1594
  initial values, 1591, 1593
Path Style Attributes dialog box, 1774, 2559
Pathline Attributes dialog box, 1822, 2582
pathlines, 1709, 1772
  accuracy control, 1776
  animation of, 1822
  coarsening, 1775
  colors, 1775
  in multiple reference frames, 614
  oil-flow, 1776
  relative, 1776
  reversing, 1776
  saving, 1777
  twisting, 1774
  XY plots along trajectories, 1776
Pathlines dialog box, 1772, 2771
PATRAN files
  exporting, 69, 82
    during a transient calculation, 86
  importing, 67, 139
  mapping data with, 97
PDF table adiabatic enthalpy, definition, 1981
PDF Table dialog box, 1068, 2240
PDF table heat loss/gain, definition, 1981
Peng-Robinson equation, 538
perforated plates, 261, 274
periodic boundaries, 130, 214, 386, 1811
  creating
    conformal, 214
    non-conformal, 164
  mesh checking, 192
  non-conformal, 153
  slitting, 215
Periodic Conditions dialog box, 584, 2468
Periodic dialog box, 387, 585, 2427
periodic flows, 582
  beta calculation, 585
  inputs for density-based solvers, 585
  inputs for pressure-based solver, 584
  limitations, 583
  postprocessing, 586
  pressure gradient, 586
  setting up parameters, 585
periodic heat transfer, 912
  constant flux, 913
  constant temperature, 913
  constraints, 912
  inputs for, 914
  postprocessing, 916
  restrictions, 912
  solution strategies, 915
  theory, 913
periodic repeats interface option, 154, 164
permeability, 262
perspective, 1808
perturbed flow, 373
Peters
  flame speed model, 1093
phase
  defining, 1377
    for Eulerian model, 1456
    for mixture model, 1447
    for VOF model, 1436
    granular, 1448, 1457, 1484
    interfacial area concentration, 1451, 1461
  diameter, 1448, 1456, 1964
  drag (mixture), 1453
  interaction, 1463
Phase Interaction dialog box, 1438, 1453-1454, 1473, 2363
phase localized compressive scheme, 1443
phase localized discretization, 1371
Phase Properties dialog box, 1378
Phases dialog box, 1377, 2355
picture
  files for animation, 1653, 1828
  options, 102
  saving files, 102
  using gray-scale colormap, 1791
Picture Options dialog box, 2776
Piecewise Linear dialog box, 1110
Piecewise-Linear Profile dialog box, 476, 2319
Piecewise-Polynomial Profile dialog box, 478, 1195, 2319
PISO algorithm, 1562
  under-relaxation, 1562
piston pin offset, 690
pixelation, 864
planar sector, 631
Plane Surface dialog box, 1722, 2777
plane surfaces, 1709, 1722
  using the plane tool, 1724
plane tool, 1723-1724
plasma-enhanced surface reaction modeling, 573
Playback dialog box, 1650, 2571
plot interpolated data, 1847
Plot Interpolated Data dialog box, 2601
Plot Profile Data dialog box, 2601
plot/circum-avg-axial , 1848
plot/circum-avg-radial , 1848

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plot/Modify Input Signal dialog box, 1877, 2605
PLOT3D files
  importing, 67
plots
  axes in, 1852
  external data, 1844
  histogram, 1839, 1851
  logarithmic, 1854
  residual, 1851
  solution, 1840, 1851
  titles for, 1845
  types of, 1838
  XY, 1838, 1840, 1844, 1846, 1851
    along pathline trajectories, 1776
    axis attributes, 1852
    circumferential average, 1848
    curve attributes, 1854
    file format, 1850
    profiles, 1846
    turbomachinery, 1871
Plots task page, 2593
  FFT, 1875
  File, 1844
  Histogram, 1851
  Profiles: Interpolated Data, 1846
  Profiles: Profile Data, 1846
  XY Plot, 1840
Point Surface dialog box, 1716, 2780
point surfaces, 1709, 1716
  using the point tool, 1717
point tool, 1716-1717
pollutant formation, 1044, 1053, 1153
  decoupled detailed chemistry model, 1202
  NOx, 1053, 1153
  soot, 1184
  SOx, 1171
polyhedra
  adaption, 203
  advantages, 199
  converting cells with hanging nodes / edges to, 143, 205
  converting domain to, 199
    boundary layer treatment, 199
    preserving interior surfaces, 203
  converting skewed cells to, 204
  exporting, 69
  face-node connectivity, 128
  hanging nodes / edges, 143, 205
  limitations, 203-205, 1684
Polynomial Profile dialog box, 475, 2318
porosity
  user-defined, 277
porous deposition
  rate, 1982
porous jump, 403
  solar load model, 903
Porous Jump dialog box, 404, 903, 2427
porous media, 261
  1D, 403
    solar load model, 903
  anisotropic inertial resistance, 270
  enabling reactions in, 269
  equations
    Darcy's Law, 263
    energy, 265
    inertial losses, 264
    thermal conductivity, 265
    transient scalar, 267
  equilibrium thermal model, 277
    equations, 265
  heat transfer, 277
  heat transfer in, 265
  inertial resistance coefficients, 270
  inputs for, 267, 273
  laminar flow inputs, 275
  limitations of, 261
  momentum equations for, 262
  moving reference frame, 261
  multiphase, 284
  multiphase flow, 280
  non-equilibrium thermal model, 277
    equations, 265
    limitation, 277
    postprocessing, 280, 1979
  packed beds, 274
  physical velocity formulation, 283
  postprocessing for, 286, 1761, 1780, 1844
  power-law model, 262, 276
    defining porosity, 277
  relative permeability, 280
  relative velocity resistance formulation, 269
  relative viscosity, 280
  solution strategies, 286
  turbulence in, 266
  UDFs
    inertial resistance, 270
    viscous resistance, 270
  velocity formulation, 261, 269, 1553
  viscous resistance coefficients, 270
positivity rate limit, 1588
postprocessing, 1747
  acoustic sources, 1222
  adjoint solver results, 2101
  Fast Fourier Transform (FFT), 1873

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

acoustic sources, 1222
FW-H acoustics model, 1219
overset meshes, 187
pollutants, 1202
pressure signals, 1222
reports, 1859, 1885
shell conduction, 843
thin walls, 373
turbomachinery, 1856
Potential dialog box , 2300
powell optimizer, 2124
power law, 489
index, 495
non-Newtonian, 493
Power Law dialog box, 490, 2323
Prandtl number, 804, 1969, 1979
pre-exponential factor, 979
PreBFC mesh files, 136
reading, 44
structured, 68, 136
unstructured, 137
precision, 1624
preconditioning, 1982
premixed
Peters
constants, 1093
Zimont
constants, 1092
premixed flames, 1089
premixed model
C-equation, 1092
G-equation, 1092
premixed turbulent combustion, 1089
boundary conditions, 1094
inputs for, 1090
model compatibility, 2847
physical properties, 1093
postprocessing, 1096
progress variable, 1094
restrictions, 1090
species concentrations, 1090, 1098
pressure
absolute, 531
coefficient, 1924, 1982
definition, 1982
drop, 396
heat exchanger, 941
dynamic, 1969
gauge, 531
interpolation schemes, 1556
jump
2D fan, 392
3D fan, 289
exhaust fan, 356
intake fan, 338
operating, 530
reduced, 1985
reference, 531, 1513
relative total, 1986
static, 1992
total, 1997
pressure boundary conditions, 308
in mixing plane model, 619
in moving reference frames, 608
in multiple reference frames, 616
pressure dependent boiling, 1251
pressure far-field boundary, 302, 347
Pressure Far-Field dialog box, 347, 2430
pressure gradient force, 1251
pressure inlet boundary, 302, 308
Pressure Inlet dialog box, 308, 2434
Pressure Outlet dialog box, 338, 2439
pressure outlets, 302, 338
non-reflecting boundary conditions, 344
targeting mass flow rate, 344
pressure signals, 1222
pressure spectrum, 1982
pressure-based coupled algorithm, 1563
pressure-based segregated algorithm
pressure-velocity coupling, 1561
pressure-based solver, 1551
coupled, 1563
density interpolation schemes, 1557
frozen flux formulation, 1612
limitations, 1551
non-iterative time advancement (NITA), 1612
porous media velocity formulation, 1553
pressure interpolation schemes, 1556
residuals, 1624
Pressure-Dependent Reaction dialog box, 980, 2339
pressure-velocity coupling
inputs for, 1561
PRESTO!, 591, 610, 835, 1556
previewing the dynamic mesh, 742
primary particle diameter, definition, 1980
Primary Phase dialog box, 1436, 2357
principal axes and principal values thermal conductivity, 506
prism layers
remeshing, 676, 678
processes, parallel, 2058
production of k, definition, 1982
production of laminar k, definition, 1982
Profile Options dialog box, 1758, 2556

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

profile plotting, 1756
  scaling factor, 1758
profiles, 243, 435
  boundary, 243, 435
    interpolation methods, 435
    reading and writing, 53
    types, 435
    units of, 111
  comma separated values, 438
  csv, 438
  excel, 438
  file format, 436
  interpolation methods, 435
    changing, 440
  plotting, 1846
  reorienting, 443
  spreadsheet, 438
  types, 435
Profiles dialog box, 440, 2385
progress variable, 1094
  definition, 1982
  sources, 301
projected surface area, 1919
Projected Surface Areas dialog box, 2615
prompt NOx parameters, 1159
prompts
  about, 30
  booleans, 31
  default values for, 34
  evaluation of, 33
  filenames, 31
  lists, 32
  numbers, 31
  strings, 31
  symbols, 31
properties, 459
  composition-dependent, 959
  database, 460, 959, 975
    data sources, 464
    user-defined, 467
  discrete phase, 1313
  for solid materials, 460
  listing, 1927
  mass diffusion coefficients, 519
  mixture, 972, 986
  modifying, 463
  non-gray model refractive index, 869
  radiation, 515
    absorption coefficient, 516
    defining, 868
    non-gray model, 869
    refractive index, 518
    reporting, 519
    scattering coefficient, 517
  saving, 466
  species, 987
  temperature-dependent, 475
    solution techniques for, 1669
    units for, 111, 475
pseudo transient, 1601
  calculation, 1605
  inputs for, 1602
  solution controls, 1604
pseudo-plastics, 494
pseudo-transient
  volume of fluid (VOF) model, 1409
PTC Mechanica Design files
  importing, 67
  study, 67
pull velocity, 2003
  inputs for, 1530
pyramidal cells, 126
pyrolysis, 1322

Q

Q criterion, definition, 1983
Quadratic of Mixture Fraction dialog box, 1110
Quadric Surface dialog box, 1726, 2781
quadric surfaces, 1709, 1726
quadrilateral cells, 123
quadrupoles, 1206
quality
  meshes, 131
    checking, 133
    improving, 194
    mesh morpher/optimizer, 2125
quality-based smoothing, 231
quantities
  data file, 107

R

radial distribution, 1450, 1460, 2360, 2363
radial equilibrium pressure distribution, 339
radial pull velocity, definition, 1984
radial velocity, 319
  definition, 1984
radiation
  boundary conditions, 869
radiation boundary temperature correction, 869
radiation heat flux, 1911
  definition, 1984
radiation Intensity Normalized Standard Deviation
  definition, 1984
Radiation Model dialog box, 845, 851, 865, 879, 2190

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

radiation models
about, 845
how to use, 845
material properties, 868
Monte Carlo (MC)
boundary conditions, 876
procedure for setting up, 845
S2S
boundary conditions, 870
reporting values, 885
radiation properties, 515
refractive index, 518
reporting, 519
radiation temperature, definition, 1984
radiative heat transfer
black body temperature, 869
boundary conditions, 869
inlets and exits, 869
walls, 870-871
discrete phase, 1250, 1324
discrete transfer radiation model (DTRM)
clustering, 883
ray tracing, 883
emissivity, 869
inputs for, 845
material properties, 868
modeling, 845
Monte Carlo (MC) radiation model
inputs for, 867
non-gray discrete ordinates (DO) model
inputs for, 865
non-gray discrete ordinates (DO) radiation model
absorption coefficient, 869
refractive index, 869
non-gray P-1 radiation model
absorption coefficient, 869
inputs for, 847
refractive index, 869
participation in radiation, 877
properties, 515
reporting, 882
solar calculator, 891
solar load model, 886
solution, 878, 880
surface-to-surface (S2S) radiation model
reporting, 885
turning on, 845
Radiator dialog box, 398, 2445
radiators, 396
RadTherm files
exporting, 69, 83
during a transient calculation, 86
rake surfaces, 1709, 1718
using the line tool, 1720
RAMPANT case and data files, 53
RAMPANT case files, 144
random wave boundary, 1419
RANS turbulence models, 752
RANS/LES Interface dialog box, 2446
raster file, 105
rate of coagulation, definition, 1983
rate of fuel NO, definition, 1984
rate of N2OPath NO, definition, 1983
rate of NO, definition, 1983
rate of nucleation, definition, 1983
rate of nuclei, definition, 1983
rate of oxidation, definition, 1983
rate of prompt NO, definition, 1983
rate of reburn NO, definition, 1983
rate of SNCR NO, definition, 1983
rate of soot mass nucleation, definition, 1983
rate of soot, definition, 1983
rate of surface growth, definition, 1983
rate of thermal NO, definition, 1983
rate of user NO, definition, 1984
ray file, 850
ray tracing, 848, 850
Rayleigh number, 1608
Rayleigh-number flows, 837
raytracing_acc , 2064
Reacting Channel 2D Curves dialog box, 2276
reacting channel model, 1012
inputs for, 1013
Reacting Channel Model dialog box , 2274
reacting flows, 957
CHEMKIN-CFD solver, 963
eddy-dissipation concept, 961
eddy-dissipation model, 961, 1380
finite-rate model, 961, 1380
inputs for, 960
overview of inputs for, 958, 1013
partially premixed combustion, 1099
pollutant formation in, 1153
premixed combustion, 1089
solution techniques, 1670
stiff chemistry solver, 963
Reaction Mechanisms dialog box, 984, 2342
reaction progress variable, 1094
reaction rate, 1998
definition, 1984
reactions
defining, 977
for fuel mixtures, 983
zone-based mechanisms, 984, 988

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

reversible, 987
Reactions dialog box, 2334
reactor net mass fraction of species-n, 1985
reactor net temperature, 1985
reactor net zone ID, 1985
reactor network
reactor net mass fraction of species-n, 1985
reactor net temperature, 1985
reactor net zone ID, 1985
Reactor Network dialog box , 2271
reactor network model, 1020
limitations, 1021
postprocessing, 1024
solving, 1021
Read Mesh Options dialog box, 44, 2634
reading
boundary conditions, 56
boundary profiles, 53
case files, 45-47
multiple, 145
core porosity model parameters, 942
data files, 45, 47
multiple, 145
parallel, 52
DTRM Ray files, 850
files, 39
compressed, 41
shortcuts, 39
using toolbar buttons, 44
injections, 1286
meshes, 44, 115, 135
multiple, 145, 212
parallel data files, 52
profiles
boundary, 53
surface meshes, 149
view factors, 863
XY files, 22
real gas model
NIST
limitations, 551
real gas models
choosing, 534
Cubic Equation of State, 532
Aungier-Redlich-Kwong, 532, 537
Peng-Robinson, 537
Redlich-Kwong, 537
Soave-Redlich-Kwong, 537
NIST, 532, 553
binary saturation tables, 558
full look-up tables, 555
multi-species, 557
user defined model (UDRGM), 561
user-defined model (UDRGM), 532
user-defined model UDRGM
example, 569
realizable k-epsilon model, model compatibility, 2847
recursive projection method (RPM), 1578
redistribute-boundary-layer , 1698
Redlich-Kwong equation, 535, 537
reduced pressure
definition, 1985
reduced temperature, definition, 1985
reference frames, 1263
multiple, 612
single, 605
Reference Mass Fractions dialog box, 2341
reference pressure
actual location, 532
location, 531, 1513
reference temperature, 987
reference values, 1924
setting general, 1925
setting reference zone, 1927
Reference Values dialog box, 1925
Reference Values task page, 624, 639, 2502
reference zone, 624, 639, 1766
reflected IR solar flux, definition, 1985
reflected radiation flux, definition, 1985
reflected visible solar flux, 1985
REFPROP database, 551
refractive index, 518
definition, 1985
non-gray radiation, 518
refrigerant, 532
region adaption, 1692
Region Adaption dialog box, 1693, 2782
registers, 1700, 1736
boundary, 1738
combining, 1701
deleting, 1701, 1703
displaying, 1704
field variable, 1739
manage, 1743
manipulating, 1701
modifying, 1703
operation, 1744
patching with, 1591, 1595
region, 1737
residuals, 1740
types, 1700
variable limiter, 1739
volume, 1741
Yplus/Ystar, 1742

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

relative humidity, definition, 1985
relative Mach number, definition, 1986
relative permeability, 280
relative total pressure, definition, 1986
relative total temperature, definition, 1986
relative velocity, 608, 611, 616, 619, 624, 639, 1592, 1594, 1598, 1766, 1933
  angle, 1986
  definition, 1985
  magnitude, 1987
relative velocity formulation, 609
  in multiple reference frames, 614
relative velocity resistance formulation, 261
relative viscosity, 280
Relaxation Options dialog box, 2508
remeshing
  additional local, 670
remeshing methods, 665
  2.5D surface, 685
  cell zone, 676
  CutCell zone, 681
  face region, 676
    with prism layers, 678
  local, 668
    using size functions, 670
  local cell, 669
  local face, 669
remote execution
  hiding graphics windows, 1783
remote shell (Linux), 2029
Rename Collision Partner dialog box , 2287
Rename Constraint dialog box, 2112
Rename dialog box, 2630
rendering options, 1798
reordering the domain, 226
reorienting profiles, 443
repairing meshes, 194
Replace Cell Zone dialog box, 218, 2785
replacing
  meshes, 222
  zones, 218
report definition, 1885
report definitions, 1885
Report Definitions dialog box, 2785
report file, 1885
Report File Definitions, 1905
Report File Definitions dialog box, 2787
report plot, 1885
Report Plot Definitions, 1907
Report Plot Definitions dialog box, 2788
report/fluxes/film-mass-flow , 1545
report/fluxes/mass-flow , 1524
reporting, 1885
  adjusting integral quantities, 1885
  case settings, 1927
  center of pressure, 1916
  conventions, 1885
  cpu usage, 1927
  creating surfaces for, 1709
  data, 1885
  drag coefficients, 1916
  electric potential, 1549
  fluxes through boundaries, 1911
  forces, 1916
  gpu information, 1927
  heat flux, 1911
  histograms, 1924
  lift coefficients, 1916
  mass flux , 1911
    for Eulerian wall film, 1545
  memory usage, 1927
  moments and moment coefficients, 1916
  optimization history for the mesh morpher/optimizer, 2125
  options for velocity, 1933
  output parameters, 1909
  parameters, 1909
  poor quality elements, 194
  projected surface areas, 1919
  radiation heat flux, 1911
  reference values, 1924
  solver time, 1927
  summary reports, 1927
  surface integrals, 1920
  turbomachinery quantities, 1859
  volume integrals, 1922
reporting soot quantities, 1201
Reporting Variables dialog box, 2562
reports
  mesh check, 193
Reports task page , 2611
resetting
  diesel unsteady flamelet, 722
  diesel unsteady flamelets, 1041
  inert EGR, 722
resetting data, 1601
residence time, 1343
Residual Monitors dialog box, 1627, 2789
residual reduction rate criteria, 1578
residual smoothing, 1577
residuals, 1987
  adjoint solver, 2100
  definition of
    density-based solver, 1626

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

pressure-based solver, 1624
display controls, 1632
display of, 1627
divergence of, 1564
for the discrete ordinates (DO) radiation model, 881
for the discrete transfer radiation model (DTRM), 881
for the P-1 radiation model, 881
for the surface-to-surface (S2S) radiation model, 881
monitoring, 1624
normalization of, 1625-1626, 1628
plotting, 1851
reduction of, 1668
renormalization of, 1628
scaling of, 1624, 1626
XY plots, 1851
resistance coefficients, 270
user-defined, 270
Results task page , 2553
reversible reactions, 987
Reynolds number, 1911, 1924, 1962
Reynolds Stress models (RSM), 753
Reynolds stress models (RSM)
model compatibility, 2847
solution strategies, 812
Reynolds stresses, 1999
Ribbon Attributes dialog box, 1332, 1774, 2560
Ribbon Reference Guide, 2633
rich flammability limit option, 1035
rich limit, 1035
Riemann invariants, 349
rigid body motion, 726
six DOF solver, 707
RMS
custom field functions, 1623
flow quantities, 1623
mass fraction, 1987
RMSE
custom field functions, 1623, 1987
flow quantities, 1623
solution variable, 1987
RNG k-epsilon model, model compatibility, 2847
rolling friction collision law, 1257
root-mean-square flow quantities, 1623
root-mean-square-error flow quantities, 1623
rosenbrock optimizer, 2125
Rosin-Rammler size distribution, 1281
Rosseland radiation model
material properties, 868
properties, 515
setting up, 845
Rotate Mesh dialog box, 229, 2792
rotating
meshes, 229
objects in a scene, 1820
views, 1800, 1805
rotating flows, 587-588, 610
mesh sensitivity in, 589
moving reference frame, 588
overview, 588
pressure interpolation, 1556
three-dimensional, 588
rotating reference frame
patching values, 1594
postprocessing, 1927
solution initialization, 1592, 1598
rotation axis, 255, 260, 605-606, 616, 619
rothalpy, definition, 1987
rough wall model, 1292
rough walls, 364, 368, 807
round-off error, 1624
rsh (Linux), 2029
Rubinow-Keller, 1291
Run Adjoint Calculation dialog box, 2101
Run Calculation task page, 1600-1601, 2537

S

S2S Information dialog box, 885, 2792
S2S radiation model
boundary conditions, 870
reporting, 885
setting up, 851
surface cluster
defining automatically, 855
defining manually, 853
setting parameters, 853
surface clusters
controlling, 853
view factor calculation, 856
view factors
computing, 856-857, 860
reading, 863
setting parameters, 853
Saffman's lift force, 1251
Sample Trajectories dialog box, 1355, 2622
Sampling Options dialog box, 2546
SAS turbulence model, 758
Save Output Parameter dialog box, 2631
Save Picture dialog box, 102, 2568
saving
boundary conditions, 56
case and data files, 47, 206
case files, 46
data files, 47
options for, 107

parallel, 52
files, 39
compressed, 41
shortcuts, 39
using toolbar buttons, 44
model settings, 56
parallel data files, 52
pathlines, 1777
EnSight format, 1779
Fieldview format, 1778
GAMBIT format, 1778
picture files, 102
solver settings, 56
SBES turbulence model, 759
setting up, 805
Scale Mesh dialog box, 226, 2162
Scale-Adaptive Simulation (SAS) turbulence model, 758
curvature correction, 798
intermittency transition model, 800
Scale-Resolving Simulation (SRS) turbulence models, 756
scaling
meshes, 226
objects in a scene, 1820
vectors, 1766
views, 1805
scattering
coefficient, 517, 868, 1988
phase function, 518
scattering coefficient, 517
constant, 518
scattering phase function
Delta-Eddington, 518
inputs for, 518
isotropic, 518
linear-anisotropic, 518
user-defined, 518
scene
creation, 1781
display, 1780
overlay, 1781
scene description, 1815
Scene Description dialog box, 1785, 1815, 2577
Scheme, 27, 29, 31, 35
executing functions in the .fluent file, 108
loading source files, 57
Scheme procedure
input parameters, 247
Schmidt number, 527, 804, 986
SDES turbulence model, 759
setting up, 805
search pattern, 21
second-order accuracy, 1555
second-order time stepping, 638
secondary mixture fraction
mean, 1988
variance, 1988
Secondary Phase dialog box, 1437, 1447-1448, 1451, 1456-1457, 1461, 2357
secondary stream, 1033
solution parameters, 1084
secure shell (Linux), 2029
segregated algorithm
pressure interpolation schemes, 1556
Select File dialog box, 146
Select Input Parameter dialog box, 2384
selective multigrid solver, 1578
semi-transparent walls, 871, 873
Separate Cell Zones dialog box, 211, 2793
Separate Face Zones dialog box, 209, 2794
separated flows, 133
separated fluid regions, 827
separating
cell zones, 208, 210
face zones, 208
Set Dual Cell Heat Exchanger dialog box, 2202
Set Injection Properties dialog box, 1049, 1285-1286, 2796
Set Multiple Injection Properties dialog box, 1297, 2803
Set Spark Ignition dialog box, 2244
Set Units dialog box, 112, 2166
sets of six DOF properties, 707
SGE
job scheduling, 2015
shadow nodes
duplicate, 194
shallow wave inputs, 1424
shape optimization, 2067
adjoint solver, 2067
mesh morpher/optimizer, 2123
shear layers, 133
shear stress, 133, 381, 1960, 2002-2003
laminar flow, 382
shear thinning, 496
shell conduction, 376
about, 838
file format, 2868
initialization, 1593
limitations, 839
locking the temperature, 297
managing, 840
physical treatment, 838
postprocessing, 843
setting up, 226

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

with the S2S radiation model, 851
Shell Conduction Layers dialog box, 2804
Shell Conduction Manager dialog box, 2806
Shielded Detached Eddy Simulation (SDES) turbulence model, 759
curvature correction, 798
intermittency transition model, 800
setting up, 805
shielding function, 805
postprocessing, 1990
shock waves, 133, 1684
postprocessing for, 1761, 1780, 1844
SIMPLE algorithm, 1562
comparison to SIMPLEC, 1562
simple effectiveness model, 920
features, 930
restrictions, 931
SIMPLEC algorithm, 1562
under-relaxation, 1562
simplex optimizer, 2124
single moving reference frame, 605
single phase flows
UDS transport equations, 576
Single Rate Devolatilization Model dialog box, 1321, 2350
single reference frame
model compatibility, 2847
single-precision solvers, 1624
Site Parameters dialog box, 985, 2344
site species, 984
Six DOF Properties dialog box, 707, 2482
six DOF solver, 643, 705
one DOF motion, 707
rigid body motion, 707
sets of properties, 707
settings, 705
size functions
local remeshing using, 670
skewed tetrahedral cells, 204
skewness, 129, 134
equiangular, 1961
equivolume, 1961
reported during mesh preview, 743
skin friction coefficient, 1924
definition, 1991
sliding interfaces, 633
sliding meshes , 628
boundary conditions, 634
cell zone conditions, 634
constraints, 631, 633, 637
contour display, 639
file saving, 634, 637
initial conditions for, 613
mesh interface shapes, 631
mesh requirements, 633
mesh setup, 630
model compatibility, 2847
patching values, 1594
postprocessing, 639, 1766, 1927
rotation speed, 634
setup, 634
solution initialization, 1592, 1598
solution procedure, 637
time step, 637-638
translational, 634
slip wall, 358, 361-362, 384
slitting
face zones, 215
periodic boundaries, 215
slope limiter, 1590
filter, 1590
smoothing, 230, 644
boundary layer, 658, 737
diffusion-based, 650
applicability, 655
diffusivity based on boundary distance, 653
diffusivity based on cell volume, 655
setting up, 644
from a reference position, 657
Laplacian, 231, 657
setting up, 644
linearly elastic solid based, 656
applicability, 656
quality-based, 231
setting up, 644
skewness-based, 234
spring-based, 646
setting up, 644
SNCR parameters, 1164
Soave-Redlich-Kwong equation, 538
solar calculator, 891
inputs/outputs, 891
theory, 892
Solar Calculator dialog box, 2199
solar heat flux, definition, 1991
solar load model
applications, 887
boundary conditions, 899
discrete ordinates (DO) irradiation, 890
boundary conditions, 905
postprocessing, 909
animation, 910
serial solver, 894
setup, 894

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

graphical user interface, 894
GUI, 894
solar calculator, 891
solar ray tracing, 887
boundary conditions, 900
glazing materials, 888
inputs, 889
shading algorithm, 888
text interface commands, 907
TUI
additional commands, 909
adjacent fluid cells, 908
align camera, 908
autoread solar data, 907
autosave solar data, 907
commands, 907
ground reflectivity, 909
quad tree refinement, 908
scattering fraction, 908
user-defined functions (UDFs), 894
solar ray tracing, 887
boundary conditions, 900
glazing materials, 888
inputs, 889
shading algorithm, 888
solid (coal) fuel NOx parameters, 1161
Solid dialog box, 259, 2379
solid materials, 260, 460, 999, 1002
solid species, 974, 984
solid suspension, 1238
solid time step, 829
solid zone, 259
locking the temperature, 297
solidification and melting, 1527
in a VOF calculation, 1447
inputs for, 1527
postprocessing, 1532
solution procedure, 1532
Solidification and Melting dialog box, 1527, 2288
solids pressure, 1450, 1460
solidus temperature, 1527
solutal buoyancy
including, 1531
solution, 1551
accuracy, 1555
animating, 1647
calculations, 1600-1601
convergence, 1624
convergence and stability, 1667
executing commands during, 1638
gradient limiters, 1590
filter, 1590
histograms, 1851
initialization, 1591, 1593, 1601
inputs, 1551
interpolation, 94
for fluid-structure interaction (FSI) problems, 97
interrupt of, 1601
limits, 1586
pressure, 599
temperature, 599, 827
monitoring, 1633
non-iterative solver (NITA), 1566
parameters
listing, 1927
non-iterative solver, 1566
pseudo transient, 1602
under-relaxation, 1564
procedure, 1551
process, 1551
pseudo transient, 1602, 1604
stability, 1564
techniques, 1674
compressible flow, 599
discrete phase, 1325
for convergence monitoring, 1633
for heat transfer, 828
for periodic heat transfer, 915
for porous media, 286
for radiation, 878
for reacting flows, 990, 1670
for swirling or rotating flows, 591
for turbulence, 810
step-by-step, 1669
turning equations on/off, 1669
under-relaxation, 1564
XY plots, 1840
Solution Controls task page, 600, 1564, 1566, 1572, 1576-1577, 1603, 2508
solution data
exporting, 69
after a calculation, 70
during a transient calculation, 86
limitations, 69
mapping, 97
solution file management, 93
Solution Files dialog box, 93, 2651
Solution Initialization task page, 1591, 2520
Solution Limits dialog box, 1586, 2511
Solution Methods task page, 610, 1559, 1563, 2504
solution mode, 1
reading multiple mesh files, 146
Solution Setup task page, 2159
Solution Steering dialog box, 2541

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution task page , 2504
Solution XY Plot dialog box, 1840, 2595
Solution Zones dialog box, 2830
solution-based adaption, adjoint, 2078
Solve VOF Every Iteration, 1374
solve/dpm-update , 1328
solve/initialize/fmg-initialization , 1596
solve/initialize/init-flow-statistics , 813
solve/initialize/set-fmg-initialization , 1596
solve/set/expert , 1632
solve/set/lock-solid-temperature? , 297
solve/set/multi-stage , 1589
solve/set/poor-mesh-numerics/cell-quality-based? , 1673
solve/set/poor-mesh-numerics/enable? , 1673
solve/set/poor-mesh-numerics/print-poor-elements-count , 1673
solve/set/poor-mesh-numerics/reset-poor-elements? , 1673
solve/set/poor-mesh-numerics/set-quality-threshold , 1673
solve/set/poor-mesh-numerics/user-defined-on-register , 1673
solve/set/stiff-chemistry , 992
solve/set/surface-tension , 1438
solver, 1551
  case check, 1665
  convergence strategies for full multigrid (FMG), 1597
  density-based, 1551
    limitations, 1551
  formulation, 1551
  inputs for, 1551
  multigrid
    full (FMG), 1595
  parallel, 2009
  pressure-based, 1551
  saving settings to a file, 56
  segregated
    limitations, 1551
  using, 1551
  using the full multigrid (FMG), 1596
soot
  mole fraction of, 1979
soot density, definition, 1991
soot mean diameter, definition, 1991
soot model, 1184
  boundary conditions, 1201
  inputs for, 1184
  method of moments, 1196
  model compatibility, 2847
  Moss-Brookes, 1190
  Moss-Brookes-Hall, 1190
  one-step, 1185
  two-step, 1187
Soot Model dialog box, 1185, 1187, 1190, 1196, 2262
soot quantities
  reporting, 1201
soot surface area, definition, 1991
soot volume fraction, definition, 1991
Soret diffusion, 522
sound pressure data, 1219
sound pressure level, 1209, 2291
sound speed, definition, 1991
source terms
  energy, 299
  in conservation equations, 297
  mass, 299
  momentum, 299
  NOx model, 301
  P-1 radiation, 300
  procedure for defining, 298
  Reynolds stress model, 300
  turbulence, 299
    k-epsilon model, 299
    k-omega model, 300
    Spalart-Allmaras model, 300
  UDS transport equations, 301, 575
  units for, 111
  user-defined scalar, 301
sources
  defining energy, 297
    input parameters, 299
  defining mass, 297
    input parameters, 299
  defining momentum, 297
    input parameters, 299
  units for, 298
SOx model, 1171
  boundary conditions, 1181
  formation, 1171
  fuel streams, 1173
  gaseous fuel, 1175
  inputs for, 1171
  liquid fuel, 1175
  solid fuel, 1175
  under-relaxation for, 1182
  user-defined functions for, 1181
SOx Model dialog box, 1172, 2258
SOx parameters, 1175
  solid fuel, 1176
Spalart-Allmaras model
  curvature correction, 798
Spalart-Allmaras turbulence models, 752

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Spalart-Allmaras turbulence models, model compatibility, 2847
Spark Ignition dialog box, 1139, 2243
spark model
  engine ignition, 1139
  extended coherent flame model, 1141
spark-ignited engines
  knock model, 1144-1145
species, 461, 959
  adding, 975
  boundary conditions, 379, 990
  bulk, 974-976
  cell zone conditions, 990
  defining, 973
    for fuel mixtures, 983
  deleting, 976
  diffusion, 829
  mass fractions, 990
  molar concentration, 1978
  order of, 974-976, 999
  properties, 987
  removing, 976
  reordering, 976
  sources, 990, 995
  transport, 957
    inputs for, 960
    without reactions, 1011
Species dialog box, 973, 975, 2332
Species Model dialog box, 998, 1026, 1091, 2216
species transport
  modeling diffusion, 519
  multiphase, 1378
    postprocessing, 1522
  reacting channel model
    using, 1013
specific dissipation rate, definition, 1992
specific heat capacity, 513, 986-987
  composition-dependent, 515
  constant, 514
  definition, 1992
  discrete phase, 1320
  kinetic theory, 515
  temperature-dependent, 514
specific heat ratio, definition, 1992
spectrum smoothing, 1877
specularity coefficient, 361-362
speed of sound, 1991
spinodal temperature, definition, 1992
split mass flow, 352
spray modeling, 1253
spread parameter, 1281
spring collision law, 1257
spring-based smoothing method, 646, 649
  setting up, 644
spring-dashpot collision law, 1257
SRS turbulence models, 756
SSD model, 1292-1293
ssh (Linux), 2029
SST k-omega model, model compatibility, 2847
stability, 1562, 1667, 1669
  under-relaxation and, 1564
Stabilized Scheme Settings dialog box, 2094
stagnation pressure, 595
stagnation temperature, 595
standard k-epsilon model, model compatibility, 2847
standard k-omega model, model compatibility, 2847
standard state enthalpy, 528
standard state entropy, 528
Stanton number, definition, 1996
stanton-rutland wall model, 1303, 1542
starting parallel ANSYS Fluent
  on a Linux system, 2026
    using command line options, 2026
  on a Windows system
    using command line options, 2021
    using the Fluent Launcher, 2012
  parallel
    on a Windows system, 2021
startup options
  parallel
    Linux, 2026
    Windows, 2021
state files, 74, 91
static interfaces, 160
  creating, 164
static pressure, 314, 326, 339, 348
  definition, 1992
  velocity inlet, 320
static temperature, 320, 348
  definition, 1993
steady diffusion flamelet model
  setting up, 1036
steady state volume of fluid (VOF) model, 1409
steady-state dynamic meshes, 744
  additional local remeshing, 670
step-by-step solution techniques, 1669
Sticking Coefficients dialog box , 2269
stiff chemistry, 992
stochastic collision, DPM model, 1252
stochastic particle tracking, 1294
  in coupled calculations, 1328
stochastic secondary droplet model (SSD), 1293
stoichiometry, 959
Stokes-Cunningham law, 1290

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

stored cell partition, 2040
stored cell partition, definition, 1993
strain rate, definition, 1993
stream function, definition, 1993
streams
  fuel
    NOx model, 1156
    SOx model, 1173
streamwise-periodic flow, 582
Stress-Blended Eddy Simulation (SBES) turbulence model, 759
  curvature correction, 798
  intermittency transition model, 800
  setting up, 805
stretch factor, definition, 1993
stretched meshes
  convergence acceleration, 1573
Strict Conditions dialog box, 2117
strings , 31
study
  PTC Mechanica Design, 67
Sub-Time Step Calculation Method, 1374
subcritical condition, definition, 1993
subgrid dissipation rate, definition, 1994
subgrid dynamic Prandtl number, definition, 1994
subgrid dynamic Sc of species, definition, 1994
subgrid dynamic viscosity constant, definition, 1994
subgrid filter length, definition, 1994
subgrid test–filter length, definition, 1994
subgrid turbulent kinetic energy, definition, 1994
subgrid turbulent viscosity
  definition, 1994
  ratio, 1994
subsonic compressible flows, 314, 342, 594
subtest kinetic energy, definition, 1994
supersonic compressible flows, 314, 342, 594
surface acoustic power, definition, 1995
surface area (projected), 1919
surface CHEMKIN files, 995
surface clusters
  S2S radiation model, 853
    defining automatically, 855
    defining manually, 853
surface corrosion rate, definition, 1997
surface coverage, 1995
surface deposition, 1000
  rate, 1995
surface dpdt RMS, 1995
Surface FSI Mapping dialog box, 97, 2656
surface integrals, 1920
  generating report, 1920
Surface Integrals dialog box, 1920, 2616
surface integration, 1920
surface kinetic mechanism files, 995
surface meshes
  reading, 149
  writing, 56
Surface Meshes dialog box, 149, 2807
surface pressure data, 1222
surface pressure level, 1992
surface report, 1885
Surface Report Definition dialog box, 2807
Surface Report Definitions, 1889
surface tension, 363, 1452
  inputs for, 1438, 1473
surface tension coefficient, 1455
surface-to-surface (S2S) radiation model
  boundary conditions, 870
  reporting, 885
  residuals, 881
  setting up, 851
  solution parameters, 880
  surface clusters
    controlling, 853
    defining automatically, 855
    defining manually, 853
    setting parameters, 853
    view factor calculation, 856
  view factors
    basis, 856
    computing, 856-857, 860
    method, 857
    reading, 863
    setting parameters, 853
surfaces
  clipping, 1730
  creating for displaying and reporting, 1709
  deleting, 1734
  grouping, 1734
  imprint, 1714
  isodistancing, 1733
  isosurfaces, 1709, 1728
  line, 1709, 1718
  partition, 1709, 1712
  plane, 1709, 1722
  point, 1709, 1716
  quadric, 1709, 1726
  rake, 1709, 1718
  renaming, 1734
  rotating, 1733
  thin walls, 373
    postprocessing, 373
  transforming, 1732
  translating, 1733

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

uses for, 1709
zone, 1709-1710
Surfaces dialog box, 1734, 2810
Sutherland Law dialog box, 489, 2322
swapping, 230, 235
Sweep Surface dialog box, 1782, 2565
sweep surfaces, 1782
swelling, 1322
swirl number, 589
swirl pull velocity, definition, 1996
swirl velocity, 319, 588
definition, 1996
for 2D fans, 390, 394
swirling flows, 587-588
mesh sensitivity in, 589
moving reference frame, 588
overview, 588
pressure interpolation, 1556
solution strategies for, 591
swirl velocity, 588
three-dimensional, 588
turbulence modeling in, 589
symbols , 31
symmetry boundary, 384, 1811
Symmetry dialog box, 2447
symmetry in the display, 1811
system commands
about, 34
for Windows, 35
system coupling motion, 738
solution stabilization, 739
system coupling thermal conditions, 377

T

TAB model, 1292
tangential velocity, 319
definition, 1996
target mass flow rate
option, 344
settings for, 345
solution strategies for, 346
UDFs for, 347
task page, 10
Tecplot files
exporting, 69, 83
during a transient calculation, 86
importing, 67
temperature, 1986, 1993, 1997, 2002
definition, 1996
locking for solids and shells, 297
reduced, 1985
thin walls, 2002
temperature averaging coefficient, 1323
temperature-dependent properties, 475, 828
density, 484
specific heat capacity, 514
thermal conductivity, 498
viscosity, 488
termination criteria, 1578
tetrahedral cells, 124
text
prompts, 30
user interface, 27
text interface
.fluent file, 108
commands
abbreviations, 28
alias, 30
line history, 29
scheme, 29
help system, 36
interrupts, 34
menu commands, 35
menu system, 27
prompt system, 30
prompts
booleans, 31
default values, 34
evaluation, 33
filenames, 31
lists, 32
numbers, 31
strings, 31
symbols, 31
system commands, 34
for Linux, 34
for Windows, 35
text user interface, 27
thermal accommodation coefficient, 987
thermal boundary conditions
external heat transfer coefficient, 383
heat flux, 383
thermal buoyancy
including, 1531
thermal conductivity, 497, 986-987, 1969
anisotropic, 500
principal axes and principal values, 506
user-defined, 507
biaxial, 502
composition-dependent, 499
constant, 498
cylindrical orthotropic, 504
definition, 1996
discrete phase, 1323

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

kinetic theory, 499
orthotropic, 503
temperature-dependent, 498
thermal diffusion coefficients
about, 522
definition, 1996
inputs for, 523
Thermal Diffusion Coefficients dialog box, 2346
thermal diffusivity, 1093
thermal expansion coefficient, 480
thermal mixing, 825
thermal resistance, 373
thermophoretic coefficient, 1323
thermophoretic force, 1251
thin walls, 373
postprocessing, 373
third-body efficiencies, 979
Third-Body Efficiency dialog box, 979, 2338
Thread Control dialog box, 2058, 2811
threads, parallel processing, 2058
tilde expansion, 43
time step, 1616
appending to file name, 48, 87
definition, 1996
discrete phase, 1246
scale, 1996
time stepping
adaptive, 1618
explicit
restrictions, 1575
variable, 1621
time-dependent problems, 1608
adaptive time stepping, 1618
animations of, 1647
boundary conditions for, 448
inputs for, 1609
mean flow quantities and custom field functions, 1623
postprocessing, 1622, 1647
root-mean-square quantities and custom field functions, 1623
root-mean-square-error quantities and custom field functions, 1623
solution parameters for, 1616
statistical analysis, 1613, 1623
time-periodic, 638
variable time stepping, 1621
time-periodic flows, 638
titles, 1788, 1845
tmerge , 148
toolbar, 7
standard, 7, 44
tools
display options, 9
pointer, 7
projection, 9
view, 8
toothpaste, 496
topology
global setting, 1872
torczon optimizer, 2124
torque, 1863
torque converters, 622-623
total Echem heat source, definition, 1996
total energy, definition, 1997
total enthalpy, definition, 1997
total heat transfer rate, 1913
total pressure, 310, 595, 599
definition, 1997
total sensible heat transfer rate, 1913
total temperature, 310, 326, 595, 599
definition, 1997
tpoly , 143
Track Style Attributes dialog box, 1332, 2562
Trajectory Sample Histograms dialog box, 2623
transcript files, 59
Transform Surface dialog box, 1732, 2812
Transformations dialog box, 1819, 2580
transforming objects in a scene, 1819
transient boundary conditions, 448
transient particles, 1242
Translate Mesh dialog box, 228, 2813
translating
meshes, 228
objects in a scene, 1820
reference frames, 613
views, 1800, 1807
translation velocity, 606, 616, 619
transmitted radiation (IR) solar flux, definition, 1998
transmitted radiation flux, 1998
transmitted visible solar flux, definition, 1998
transonic compressible flows, 594
transparency, 1818
transparent flow forcing, 405, 425
transport equations
user-defined scalars, 573, 575
tree, 3
triad
manipulating, 1803
visibility, 1788-1789
triangular cells, 122
triangular face approach, 162
troubleshooting of calculations, 1587
Tsuji-et-al, 1291

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tube banks, 261
TUI , 27
  commands in .fluent file, 108
Turbo 2D Contours dialog box, 1869, 2814
Turbo Averaged Contours dialog box, 1868, 2816
Turbo Averaged XY Plot dialog box, 1871, 2817
Turbo menu
  2D Contours..., 1869
  Averaged Contours..., 1868
  Averaged XY Plot..., 1871
  Options..., 1872
  Report..., 1859
Turbo Options dialog box, 1872, 2819
Turbo Report dialog box, 1859, 2819
Turbo Topology dialog box, 1856, 2822
turbo-specific non-reflecting boundary conditions
  overview of, 406
  theory, 409
  using, 414
turbomachinery, 1732
  2D contours, 1869
  average flow angles, 1862
  average total pressure, 1861
  average total temperature, 1862
  averaged contours, 1867
  axial force, 1863
  coordinates, 1872
  defining topology, 1856
  efficiency, 1864, 1866, 1924
  mass flow, 1861
  passage loss coefficient, 1863
  postprocessing, 1856
  quantities, 1861
  report, 1859
  setting global topology, 1872
  swirl number, 1861
  topology, 1856
  torque, 1863
  XY plots, 1871
turbulence, 749, 1252
  boundary conditions, 807
    computing, 305
    input methods, 303
    relationships for deriving, 305
  choosing a model, 751
  compressibility effects, 755, 799
  crossflow instability, 800
  Curvature Correction for Spalart-Allmaras, 755
  delayed detached eddy simulation DDES model, 800
  Detached Eddy Simulation (DES) model, 758
  dissipation rate, 1998
  Embedded LES (ELES) model, 760
  grid considerations for, 811
  grid resolution RANS models, 756
  Grid Resolution SRS model, 760
    free shear flows, 760
    wall boundary layers, 760
  hybrid RANS-LES models, 757
  inputs for, 764
  intensity, 304, 1998
  intermittency transition model, 800
  k-epsilon models, 752
  k-omega models, 753
  kinetic energy, 1998
  laminar-turbulent transition models, 754
  large eddy simulation (LES), 757
  Large Eddy Simulation (LES) model, 757
  length scale, 304
  mesh considerations for, 1695
  model compatibility, 2847
  model enhancements, 755
  model hierarchy, 763
  modeling, 749
  near-wall treatments, 767, 769
    Werner-Wengle, 792
  options, 797
  Prandtl number, 804
  RANS models, 752
  reporting, 814
  RSM, 753
  SAS model, 758
  Scale-Adaptive Simulation (SAS) model, 758
  Scale-Resolving Simulation (SRS) models, 756
  Schmidt number, 804
  Shielded Detached Eddy Simulation (SDES) model, 759
    setting up, 805
  solution strategies, 810
  sources, 299
  Spalart-Allmaras models, 752
  SRS models, 756
  SRS numerics, 761
    convergence control, 763
    iterative scheme, 762
    spatial discretization, 762
    time discretization, 761
  Stress-Blended Eddy Simulation (SBES) model, 759
    setting up, 805
  transition, 255, 283
  troubleshooting, 824
  user-defined functions, 804
  wall roughness, 364, 368, 807
  wall treatment RANS models, 755
turbulence interaction

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Eulerian multiphase, 1476
turbulence model enhancements, 755
turbulence models
model hierarchy, 763
SRS numerics, 761
convergence control, 763
iterative scheme, 762
spatial discretization, 762
time discretization, 761
turbulence parameters, 1166, 1179
turbulence source terms
defining, 299
turbulent reaction rate, definition, 1998
turbulent Reynolds number, definition, 1998
turbulent viscosity
definition, 1999
large-scale definition, 1999
modified, 1978
ratio, 305, 1999
subgrid, 1994
small-scale definition, 1999
subgrid, 1994
twisting pathlines, 1774
Two Competing Rates Model dialog box, 1321, 2351
two-sided wall, 361, 370, 375
two-step soot formation model, 1187

U

UDF
input parameters, 249
see also user-defined functions, 243
UDF Library Manager dialog box, 2823
UDS Diffusion Coefficients dialog box, 509, 2347
UDS, overview
see also user-defined scalars, 573
unburnt mixture
physical properties, 1093
species concentration, 1090, 1098
unburnt thermal diffusivity, 529
under-relaxation
default values, 1565, 1603
discrete phase, 1328
inputs for, 1564
of density, 992
of energy, 828
of temperature, 828
with PISO, 1562
with SIMPLEC, 1562
under-relaxation of variables
density-based solver, 1575
underbody simulations, 373
underhood simulations, 158, 215, 373, 858
Ungrouped Macro Heat Exchanger dialog box, 2207
ungrouped macro heat exchanger model, 932
uniform distribution
for display, 1718, 1722
uniformity index, 1920
units, 111
conversion factors, 114
custom systems, 112
for custom field functions, 2003
for flow variables, 1957
for length, 227
restrictions, 111
unsteady diffusion flamelet model
setting up, 1036
using, 1037
zeroing species, 1055
Unsteady Flamelet Parameters dialog box, 2239
unsteady flows, 1608
in multiple reference frames, 614
unsteady statistics
sampling, 1613
Use Input Parameter for UDF dialog box, 2629
Use Input Parameter in Scheme Procedure dialog box, 2628
User Defined Report Definition dialog box, 2828
User Defined Scalar Sources dialog box, 576
user interface
graphical, 1
text, 27
user-defined anisotropic diffusivity, 513
user-defined anisotropic thermal conductivity, 507
User-Defined Database Materials dialog box, 2313
user-defined fan model, 427
User-Defined Fan Model dialog box, 429, 2824
user-defined function (UDF)
anisotropic thermal conductivity, 507
diffusivity, 513
emissivity weighting factor, 847, 865
scattering phase function, 518
User-Defined Function Hooks dialog box, 2825
user-defined functions (UDFs)
boundary conditions, 243
discrete phase, 1258, 1296
Prandtl number, 804
Schmidt number, 804
turbulence, 804
units in, 111
update of, 1601
User-Defined Functions dialog box, 2321
user-defined mass flux, 574
user-defined materials database, 467
User-Defined Memory dialog box, 2827

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

user-defined quantities
non-isotropic thermal conductivity, 277
porosity, 277
resistance coefficients, 270
user-defined real gas model (UDRGM), 532
ideal gas equation example, 569
user-defined scalars (UDS)
diffusion, 508
anisotropic, 509-510
anisotropic diffusion equation, 508
cylindrical orthotropic, 512
isotropic, 508
isotropic diffusion equations, 508
orthotropic, 511
postprocessing, 576
setting up
multiphase flow, 581
single phase flow, 576
transport equations, 573, 575
diffusion, 574
multiphase flow, 574, 581
multiphase mass flux, 574
single phase flow, 574, 576
single phase mass flux, 574
User-Defined Scalars dialog box, 576, 581, 2829
user-defined sources, 2083

V

V-cycle multigrid, 1577
V-LAN Settings dialog box, 2836
values, principal, 506
vapor pressure, 1322
vaporization
pressure, 1455
temperature, 1049, 1323
vaporization model, 1324, 2308
variable definitions, 1957
Variable Time Step Settings dialog box, 2545
variable time stepping, 1621
Vector Definitions dialog box, 1770, 2558
vector file, 105
Vector Options dialog box, 1766, 2557
vectors, 1764
colors, 1768-1769
custom, 1769
Vectors dialog box, 1765, 2831
velocity
angle, 2000
relative, 1986
axial, 319, 1960
boundary conditions, 318
Cartesian, 1933
for boundary condition inputs, 318
components, 2003
cylindrical, 1933
for boundary condition inputs, 318
definition, 2000
fixed values of, 294
local cylindrical, 318
magnitude, 2000
radial, 319, 1984
reporting options, 1933
swirl, 588, 1996
tangential, 319, 1996
Velocity Effectiveness Curve dialog box, 2210
velocity far-field boundary, 320
velocity formulation
in moving reference frames, 606
porous media, 261, 269
velocity inlet boundary, 302, 316
Velocity Inlet dialog box, 317, 2447
velocity vectors, 1764
version and release information, 26
video, 1829
Video Control dialog box, 2833
view factors
accelerating, 2064
compressed row format (CRF), 860
computing, 851, 860
accelerating, 2064
for large meshes or complex models, 860, 862
hemicube method, 857
inside ANSYS Fluent, 860
limitation, 860
outside ANSYS Fluent, 862
ray tracing method, 858
GPGPUs, 2064
reading, 863
reporting, 885
setting parameters, 853
View Factors and Clustering dialog box, 853, 2194
view last , 1803
viewfac_acc , 2064
views
centering, 1805
magnifying, 1800, 1807
modifying, 1803
rotating, 1800, 1805
saving, 1810
scaling, 1805
translating, 1800, 1807
zooming, 1807
Views dialog box, 1804, 1809, 1811, 2583
virtual mass force, 1251

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

virtual polygon approach, 162
viscosity, 487, 986-987, 1969, 1979, 1999
  Carreau model, 494
  composition-dependent, 491
  constant, 488
  Cross model, 495
  kinetic theory, 490
  non-Newtonian, 492
    Carreau model, 494
    Cross model, 495
    Herschel-Bulkley model, 495
    power law, 493
  power law, 489
    non-Newtonian, 493
  Sutherland's law, 489
  temperature-dependent, 488
  zero-shear-rate, 495
Viscous dialog box, 596
viscous dissipation, 596, 825
Viscous Model dialog box, 527, 600, 764, 2176
viscous stresses, 1589
visibility, 1817-1818
  titles, 1788
  triad, 1788
visualizing results, 1747
VOF multiphase model
  solution method
    coupled pressure-based solver, 1507
volatile fraction, 1049, 1324
volume adaption, 1694
Volume Adaption dialog box, 1694, 2838
volume fraction
  in Lagrangian discrete phase model, 1238
volume fraction, definition, 2000
Volume FSI Mapping dialog box, 97, 2653
volume integrals
  generating report, 1922
Volume Integrals dialog box, 1922, 2620
volume integration, 1922
volume mesh update procedure, 689
volume of fluid (VOF) model
  coupled level set, 1409
  donor-acceptor scheme, 1373
  explicit scheme, 1373
  geometric reconstruction scheme, 1373
  implicit scheme, 1373
  interpolation
    donor-acceptor scheme, 1373
    explicit scheme, 1373
    geometric reconstruction scheme, 1373
    implicit scheme, 1373
  jump adhesion
    inputs for, 1442
  model compatibility, 2847
  numerical beach, 1432
  open channel
    numerical beach, 1432
    recommendations, 1429
  open channel flow, 1410
    limitations, 1417
    recommendations, 1418
  open channel wave boundary condition, 1419
  phase localized
    compressive scheme, 1443
  properties, 1436-1437
  pseudo-transient, 1409
  reporting flow rates, 1524
  solidification and melting in, 1447
  solution strategies, 1512
  steady state, 1409
  steady-state calculations, 1373
  surface tension
    inputs for, 1438, 1473
  time-dependent calculations, 1373
    inputs for, 1445
  wall adhesion
    inputs for, 1393-1394, 1440, 1473
volume report, 1885
Volume Report Definition, 1891
Volume Report Definition dialog box, 2839
Volumetric Absorbed Radiation, definition, 2000
Volumetric Emitted Radiation, definition, 2000
Volumetric Smoothing, 1407
volumetric species, 961
vortex shedding, 1608
vorticity, 1974
  components, 2003
  magnitude, 2001

W

W-cycle multigrid, 1577
wall
  adhesion
    inputs for, 1393, 1440, 1473
  boundary conditions, 358
    motion, 359
    shear, 361
    thermal, 370
  convective augmentation factor, 373
  coupled, 215, 375
    interface option, 156, 164
  fluxes, 2001
  heat flux, 383
  Marangoni stress, 361, 363

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

matching
interface option, 157
motion, 359, 634
moving, 608
no-slip, 361-362
rotation, 616
roughness, 364, 368, 807, 1542
shear stress, 382, 1960, 2002
components, 2003
shell conduction, 376, 838
file format, 2868
initialization, 1593
limitations, 839
locking the temperature, 297
managing, 840
physical treatment, 838
postprocessing, 843
with the S2S radiation model, 851
slip, 361-362
specified shear, 361-362
specularity coefficient, 361-362
temperature, 2002
thin walls, 2002
thickness, 373
postprocessing, 373
translation, 616
two-sided, 361, 370, 375
wall boundary layer model, 1309
Wall dialog box, 359, 361, 368, 370, 2456
wall film
face pressure, 2001
heat flux, 2001
height, 2001
impingement mass flux, 2001
impingement model, 1542
mass, 2001
temperature, 2001
velocity magnitude, 2001
x-velocity, 2001
y-velocity, 2001
z-velocity, 2001
wall function heat transfer coefficient, definition, 2001
Wall Irradiation Flux.Normalized Std Deviation, define, 2002
wall surface reactions, 974, 995
boundary conditions, 380
heat transfer, 1000
inputs for, 998
site species, 984
solid species, 984
wall treatment RANS turbulence models, 755
Warning dialog box
for merging zones, 2841
warped-face gradient correction
WFGC, 1674
WAVE, 456
wave group inputs, 1425
WAVE model, 1292
wave spectrum, 1419
wave spectrum inputs, 1425
wedge cell
remeshing, 676, 678
wedge cells, 125
weighted-sum-of-gray-gases model (WSGGM), 516
Werner-Wengle near-wall treatment, 792
wet steam multiphase model
postprocessing, 1523
properties, 1495
solving, 1518
UDF, 1496
using, 1495
wildcards, 241
window dumps, 106
windows, 3
Windows systems
graphics window features, 5
starting ANSYS Fluent on, 2021
Workbench
automatically saving files within, 51
using input parameters within, 243, 2125, 2627
Working dialog box, 18
workpile, 1360
Write Profile dialog box, 54, 2388
Write Views dialog box, 1810, 2584
writing boundary meshes, 56
WSGGM User Specified dialog box, 516, 2348

X

XML template files, 2025
XY plot files
units in, 111
XY plots, 1838, 1840, 1844, 1846, 1852
along pathline trajectories, 1776
axis attributes, 1852
circumferential average, 1848
curve attributes, 1854
file format, 1850
profiles, 1846
residuals, 1851
turbomachinery, 1871

Y

y*
adaption, 1695

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

y+, 2002
  adaption, 1695
yield stress, 495
Yplus/Ystar Adaption dialog box, 1695, 2842
y∗, 2002

Z

zero-shear-rate viscosity, 495
Zimont
  flame speed model, 1092
zonal discretization, 1371, 1398
zone motion
  specifying, 722
Zone Motion dialog box, 742, 2499
Zone Scale Info dialog box, 723, 731, 734, 2499
Zone Surface dialog box, 1710, 2844
zone surfaces, 1709-1710
zones
  activating, 221
  boundary, 237
  changing name of, 243
  changing type of, 239
  copying, 222
  deactivating, 220
  deforming, 722, 731
  deleting, 219
  modifying, 218
  moving, 603, 628
  replacing, 218
zooming, 1800, 1807

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.